FANUC Series 30*i*/300*i*/300*is*-MODEL A
FANUC Series 31*i*/310*i*/310*is*-MODEL A5
FANUC Series 31*i*/310*i*/310*is*-MODEL A
FANUC Series 32*i*/320*i*/320*is*-MODEL A

Common to Lathe System/Machining Center System

USER'S MANUAL

B-63944EN/02

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units.
It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.
Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

CONTENTS

1.1 DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into **Warning** and **Caution** according to their bearing on safety. Also, supplementary information is described as a **Note**. Read the **Warning**, **Caution**, and **Note** thoroughly before attempting to use the machine.

> ⚠ **WARNING**
> Applied when there is a danger of the user being injured or when there is a danger of both the user being injured and the equipment being damaged if the approved procedure is not observed.

> ⚠ **CAUTION**
> Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

> **NOTE**
> The Note is used to indicate supplementary information other than Warning and Caution.

- Read this manual carefully, and store it in a safe place.

1.2 GENERAL WARNINGS AND CAUTIONS

⚠ **WARNING**

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
2. Before operating the machine, thoroughly check the entered data.
   Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate.
   If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
4. When using a tool compensation function, thoroughly check the direction and amount of compensation.
   Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

⚠ **WARNING**

5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

6 Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit.
Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

7 The User's Manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

8 Some functions may have been implemented at the request of the machine-tool builder. When using such functions, refer to the manual supplied by the machine-tool builder for details of their use and any related cautions.

⚠ **CAUTION**

The liquid-crystal display is manufactured with very precise fabrication technology. Some pixels may not be turned on or may remain on. This phenomenon is a common attribute of LCDs and is not a defect.

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off.

Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.

To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

1.3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied User's Manual carefully such that you are fully familiar with their contents.

⚠ **WARNING**

1 **Coordinate system setting**
If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command. Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2 **Positioning by nonlinear interpolation**
When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming. Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Function involving a rotation axis**
When programming polar coordinate interpolation or normal-direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4 **Inch/metric conversion**
Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

⚠ **WARNING**

5 **Constant surface speed control**
When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

6 **Stroke check**
After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7 **Tool post interference check**
A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user. After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8 **Absolute/incremental mode**
If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

9 **Plane selection**
If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

10 **Torque limit skip**
Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

⚠ **WARNING**

11 **Programmable mirror image**
Note that programmed operations vary considerably when a programmable mirror image is enabled.

12 **Compensation function**
If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.
Before issuing any of the above commands, therefore, always cancel compensation function mode.

1.4 WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied User's Manual carefully, such that you are fully familiar with their contents.

⚠ **WARNING**

1 **Manual operation**
When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2 **Manual reference position return**
After switching on the power, perform manual reference position return as required.
If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed.
An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Manual numeric command**
When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid.
Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

4 **Manual handle feed**
In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

⚠ **WARNING**

5 **Disabled override**
If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

6 **Origin/preset operation**
Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

7 **Workpiece coordinate system shift**
Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

8 **Software operator's panel and menu switches**
Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

9 **RESET key**
Pressing the RESET key stops the currently running program. As a result, the servo axes are stopped. However, the RESET key may fail to function for reasons such as an MDI panel problem. So, when the motors must be stopped, use the emergency stop button instead of the RESET key to ensure security.

⚠ **WARNING**

10 **Manual intervention**
If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

11 **Feed hold, override, and single block**
The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

12 **Dry run**
Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

13 **Cutter and tool nose radius compensation in MDI mode**
Pay careful attention to a tool path specified by a command in MDI mode, because cutter or tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in cutter or tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

14 **Program editing**
If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

1.5 WARNINGS RELATED TO DAILY MAINTENANCE

⚠ **WARNING**

1 **Memory backup battery replacement**
When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.
Refer to the Section “Method of replacing battery” in the User’s Manual (Common to T/M series) for details of the battery replacement procedure.

⚠ **WARNING**

2 **Absolute pulse coder battery replacement**
When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.
Refer to the FANUC SERVO MOTOR αi series Maintenance Manual for details of the battery replacement procedure.

⚠ **WARNING**

3 Fuse replacement

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.

TABLE OF CONTENTS

III. OPERATION

IV. MAINTENANCE

APPENDIX

I. GENERAL

1 GENERAL

This manual consists of the following parts:

About this manual

I. GENERAL
Describes chapter organization, applicable models, related manuals, and notes for reading this manual.

II. PROGRAMMING
Describes each function: Format used to program functions in the NC language, explanations, and limitations.

III. OPERATION
Describes the manual operation and automatic operation of a machine, procedures for inputting and outputting data, and procedures for editing a program.

IV. MAINTENANCE
Describes procedures for daily maintenance and replacing batteries.

APPENDIX
Lists parameters, valid data ranges, and alarms.

NOTE

1 This manual describes the functions common to the lathe system and machining center system.
For the functions specific to the lathe system or machining center system, refer to the User's Manual (T series) (B-63944EN-1) or the User's Manual (M series) (B-63944EN-2).

2 Some functions described in this manual may not be applied to some products. For detail, refer to the Descriptions manual (B-63942EN).

3 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the Parameter Manual (B-63950EN).
Parameters are used to set functions and operating conditions of a CNC machine tool, and frequently-used values in advance. Usually, the machine tool builder factory-sets parameters so that the user can use the machine tool easily.

4 This manual describes not only basic functions but also optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

This manual describes the models indicated in the table below.
In the text, the abbreviations indicated below may be used.

Model name	Abbreviation	
FANUC Series 30*i*-MODEL A	30*i* –A	Series 30*i*
FANUC Series 300*i*-MODEL A	300*i*–A	Series 300*i*
FANUC Series 300*i*s-MODEL A	300*i*s–A	Series 300*i*s
FANUC Series 31*i*-MODEL A	31*i* –A	Series 31*i*
FANUC Series 31*i*-MODEL A5	31*i* –A5	
FANUC Series 310*i*-MODEL A	310*i*–A	Series 310*i*
FANUC Series 310*i*-MODEL A5	310*i*–A5	
FANUC Series 310*i*s-MODEL A	310*i*s–A	Series 310*i*s
FANUC Series 310*i*s-MODEL A5	310*i*s–A5	
FANUC Series 32*i*-MODEL A	32*i* –A	Series 32*i*
FANUC Series 320*i*-MODEL A	320*i*–A	Series 320*i*
FANUC Series 320*i*s-MODEL A	320*i*s–A	Series 320*i*s

NOTE

1 For an explanatory purpose, the following descriptions may be used according to the types of path control used:
- T series: For the lathe system
- M series: For the machining center system

2 Unless otherwise noted, the model names 31*i*/310*i*/310*i*s-A, 31*i*/310*i*/310*i*s-A5, and 32*i*/320*i*/320*i*s-A are collectively referred to as 30*i*/300*i*/300*i*s. However, this convention is not necessarily observed when item 3 below is applicable.

3 Some functions described in this manual may not be applied to some products.
For details, refer to the DESCRIPTIONS (B-63942EN).

Special symbols

This manual uses the following symbols:

- M

Indicates a description that is valid only for the machine center system set as system control type (in parameter No. 0983).
In a general description of the method of machining, a machining center system operation is identified by a phase such as "for milling machining".

- T

Indicates a description that is valid only for the lathe system set as system control type (in parameter No. 0983).
In a general description of the method of machining, a lathe system operation is identified by a phrase such as "for lathe cutting".

-

Indicates the end of a description of a system control type.
When a system control type mark mentioned above is not followed by this mark, the description of the system control type is assumed to continue until the next item or paragraph begins. In this case, the next item or paragraph provides a description common to the control types.

- **IP**

Indicates a combination of axes such as X_ Y_ Z_
In the underlined position following each address, a numeric value such as a coordinate value is placed (used in PROGRAMMING.).

- ;

Indicates the end of a block. It actually corresponds to the ISO code LF or EIA code CR.

Related manuals of
Series 30*i*/300*i*/300*i*s- MODEL A
Series 31*i*/310*i*/310*i*s- MODEL A
Series 31*i*/310*i*/310*i*s- MODEL A5
Series 32*i*/320*i*/320*i*s- MODEL A

The following table lists the manuals related to Series 30*i*/300*i* /300*i*s-A, Series 31*i*/310*i* /310*i*s-A, Series 31*i*/310*i* /310*i*s-A5, Series 32*i*/320*i* /320*i*s-A. This manual is indicated by an asterisk(*).

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-63942EN	
CONNECTION MANUAL (HARDWARE)	B-63943EN	
CONNECTION MANUAL (FUNCTION)	B-63943EN-1	
USER'S MANUAL (Common to Lathe System/Machining Center System)	B-63944EN	*
USER'S MANUAL (For Lathe System)	B-63944EN-1	
USER'S MANUAL (For Lathe Machining Center System)	B-63944EN-2	
MAINTENANCE MANUAL	B-63945EN	
PARAMETER MANUAL	B-65950EN	
Programming		
Macro Compiler / Macro Executor PROGRAMMING MANUAL	B-63943EN-2	
Macro Compiler OPERATOR'S MANUAL	B-66264EN	
C Language Executor OPERATOR'S MANUAL	B-63944EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-63983EN	
Network		
PROFIBUS-DP Board OPERATOR'S MANUAL	B-63994EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64014EN	
DeviceNet Board OPERATOR'S MANUAL	B-64044EN	
Operation guidance function		
MANUAL GUIDE *i* OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* Set-up Guidance OPERATOR'S MANUAL	B-63874EN−1	

Related manuals of SERVO MOTOR α*i*s/α*i*/β*i*s/β*i* series

The following table lists the manuals related to SERVO MOTOR α*i*s/α*i*/β*i*s/β*i* series

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR α*i*s series FANUC AC SERVO MOTOR α*i* series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS	B-65272EN
FANUC AC SERVO MOTOR β*i*s series DESCRIPTIONS	B-65302EN
FANUC AC SPINDLE MOTOR β*i* series DESCRIPTIONS	B-65312EN
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER β*i* series DESCRIPTIONS	B-65322EN
FANUC SERVO MOTOR α*i*s series FANUC SERVO MOTOR α*i* series FANUC AC SPINDLE MOTOR α*i* series FANUC SERVO AMPLIFIER α*i* series MAINTENANCE MANUAL	B-65285EN
FANUC SERVO MOTOR β*i*s series FANUC AC SPINDLE MOTOR β*i* series FANUC SERVO AMPLIFIER β*i* series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR α*i*s series FANUC AC SERVO MOTOR α*i* series FANUC AC SERVO MOTOR β*i*s series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR α*i* series FANUC AC SPINDLE MOTOR β*i* series PARAMETER MANUAL	B-65280EN

Any of the servo motors and spindles listed above can be connected to the CNC described in this manual. However, α*i* series servo amplifiers can only be connected to α*i* series SVMs (for 30*i*/31*i*/32*i*).
This manual mainly assumes that the FANUC SERVO MOTOR α*i* series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

1.1 NOTES ON READING THIS MANUAL

⚠ **CAUTION**

1 The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand-point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.

2 In the header field of each page of this manual, a chapter title is indicated so that the reader can reference necessary information easily.
By finding a desired title first, the reader can reference necessary parts only.

3 This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
If a particular combination of operations is not described, it should not be attempted.

1.2 NOTES ON VARIOUS KINDS OF DATA

⚠ **CAUTION**

Machining programs, parameters, offset data, etc. are stored in the CNC unit internal non-volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non-volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.

II. PROGRAMMING

1 GENERAL

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION

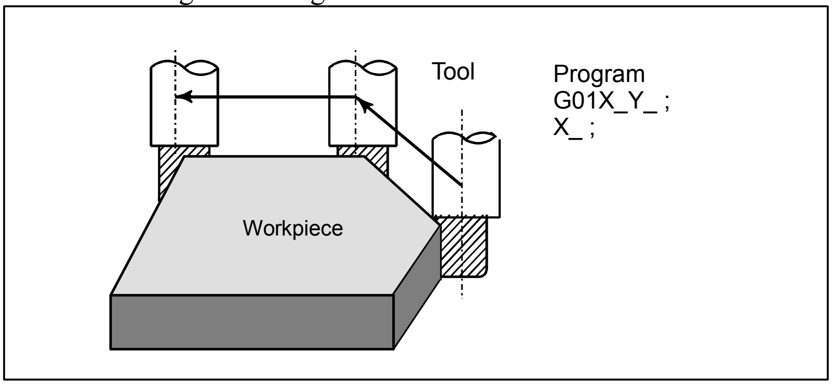
The tool moves along straight lines and arcs constituting the workpiece parts figure (See II-4).

Explanation

The function of moving the tool along straight lines and arcs is called the interpolation.

- Tool movement along a straight line

- For milling machining



- For lathe cutting

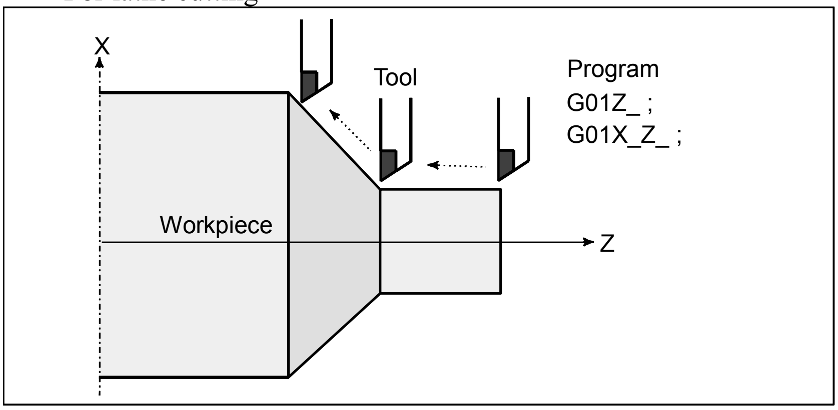
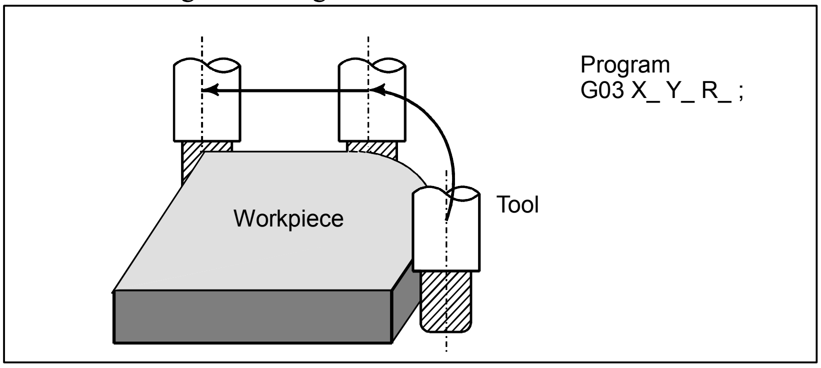


Fig. 1.1 (a) Tool movement along a straight line

- Tool movement along an arc

- For milling machining



- For lathe cutting

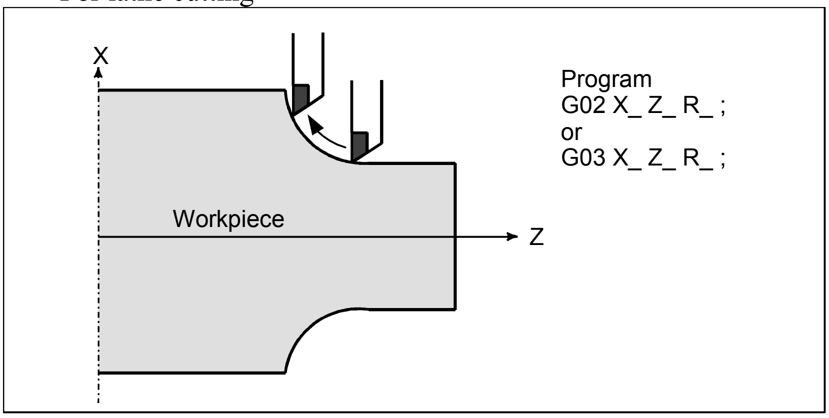


Fig. 1.1 (b) Tool movement along an arc

The term interpolation refers to an operation in which the tool moves along a straight line or arc in the way described above.
Symbols of the programmed commands G01, G02, ... are called the preparatory function and specify the type of interpolation conducted in the control unit.

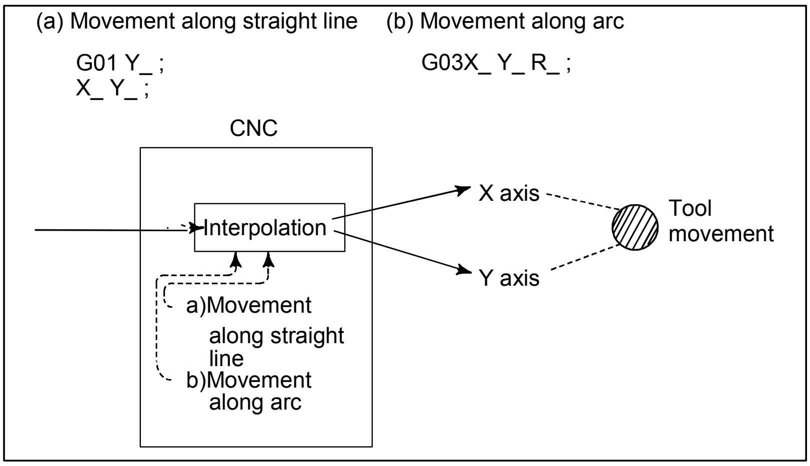


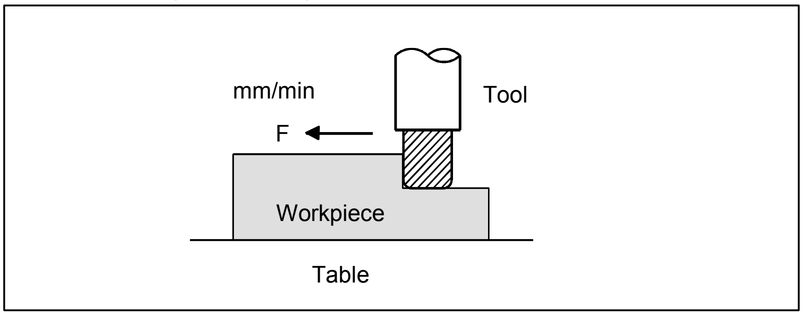
Fig. 1.1 (c) Interpolation function

> **NOTE**
> Some machines move tables instead of tools but this manual assumes that tools are moved against workpieces.

1.2 FEED-FEED FUNCTION

Movement of the tool at a specified speed for cutting a workpiece is called the feed.

- For milling machining



- For lathe cutting

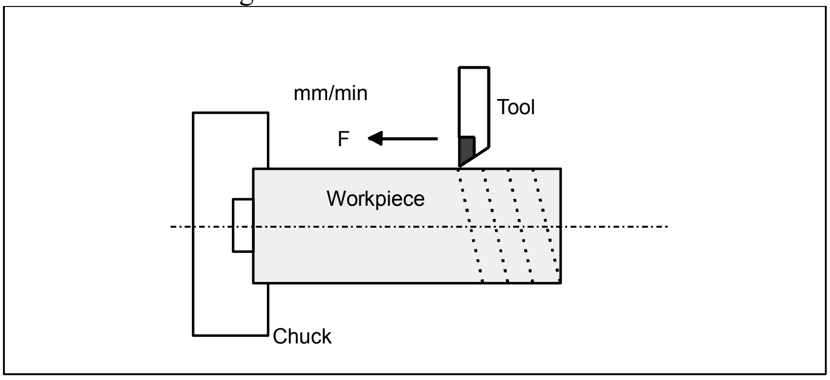


Fig. 1.2 (a) Feed function

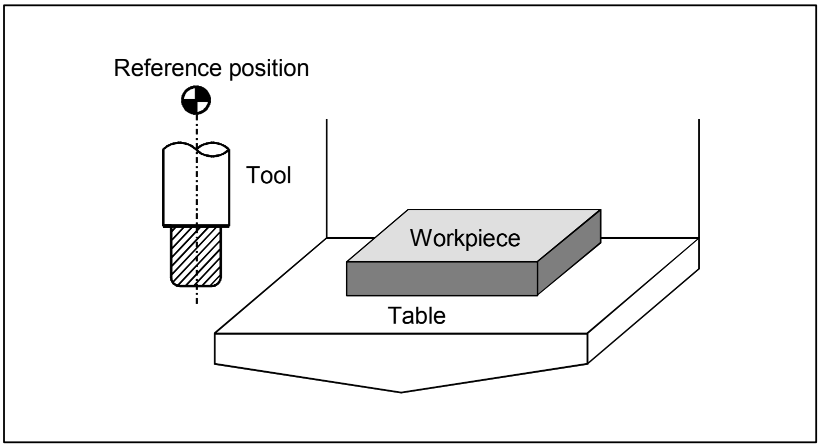
Feedrates can be specified by using actual numerics. For example, to feed the tool at a rate of 150 mm/min, specify the following in the program:
F150.0
The function of deciding the feed rate is called the feed function (See II-5).

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine-specific Position)

A CNC machine tool is provided with a fixed position. Normally, tool change and programming of absolute zero point as described later are performed at this position. This position is called the reference position.

- For milling machining



- For lathe cutting

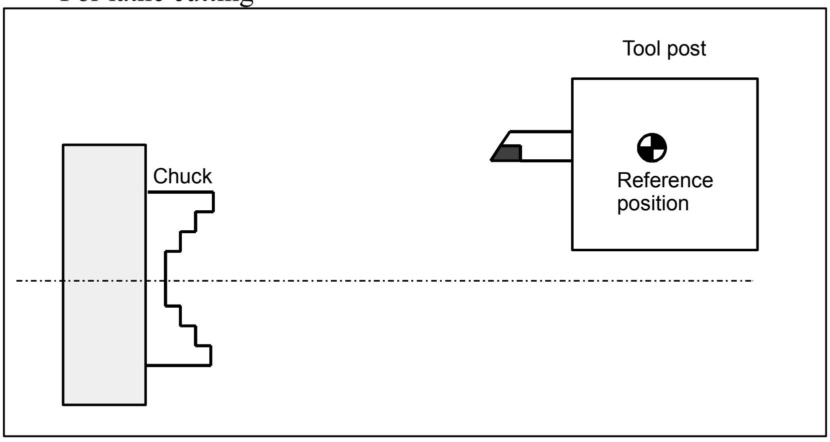


Fig. 1.3.1 (a) Reference position

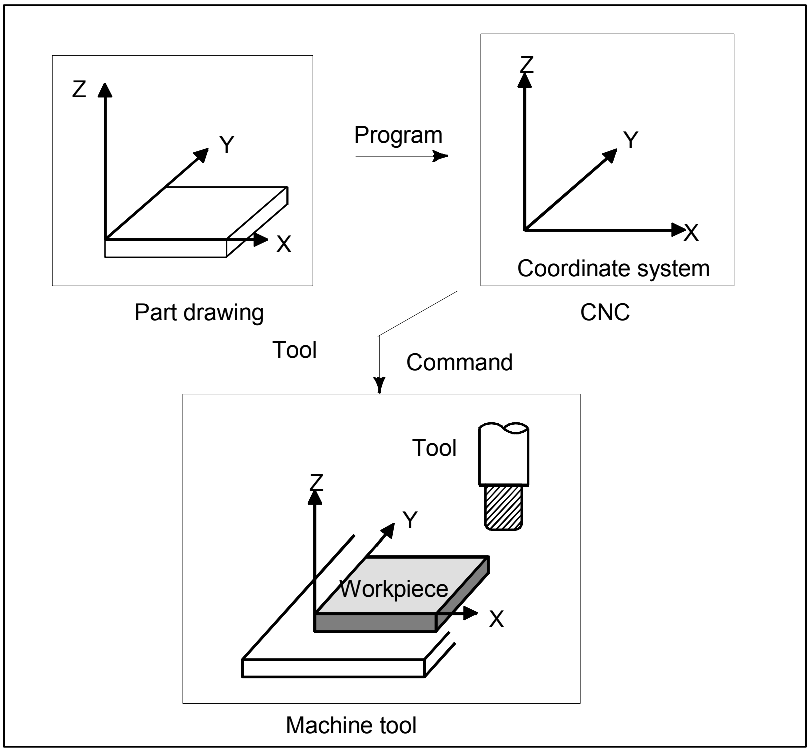
Explanation

The tool can be moved to the reference position in two ways:

1. Manual reference position return (See III-3.1)
   Reference position return is performed by manual button operation.
2. Automatic reference position return (See II-6)
   In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System

- For milling machining



- For lathe cutting

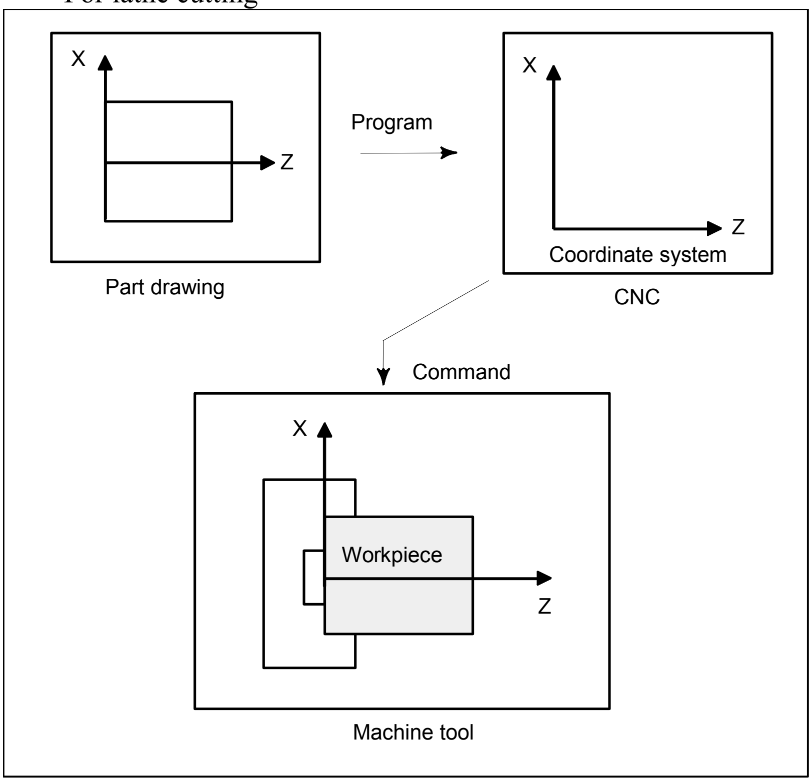


Fig. 1.3.2 (a) Coordinate system

Explanation
- Coordinate system

The following two coordinate systems are specified at different locations: (See II-7)

1 Coordinate system on part drawing
The coordinate system is written on the part drawing. As the program data, the coordinate values on this coordinate system are used.

2. Coordinate system specified by the CNC
The coordinate system is prepared on the actual machine tool table. This can be achieved by programming the distance from the current position of the tool to the zero point of the coordinate system to be set.

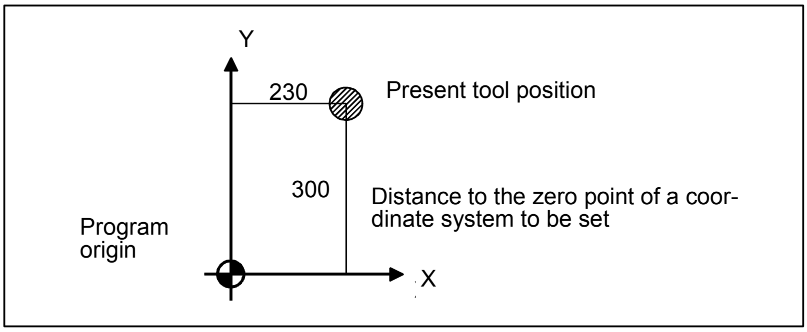
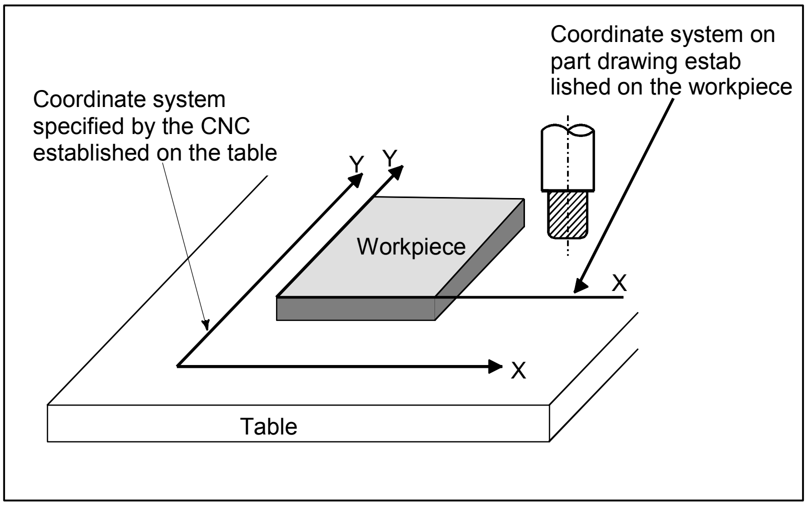


Fig. 1.3.2 (b) Coordinate system specified by the CNC

Concrete programming methods for setting coordinate systems specified by the CNC are explained in II-7, "COORDINATE SYSTEM".

The positional relation between these two coordinate systems is determined when a workpiece is set on the table.

- For milling machining



- For lathe cutting

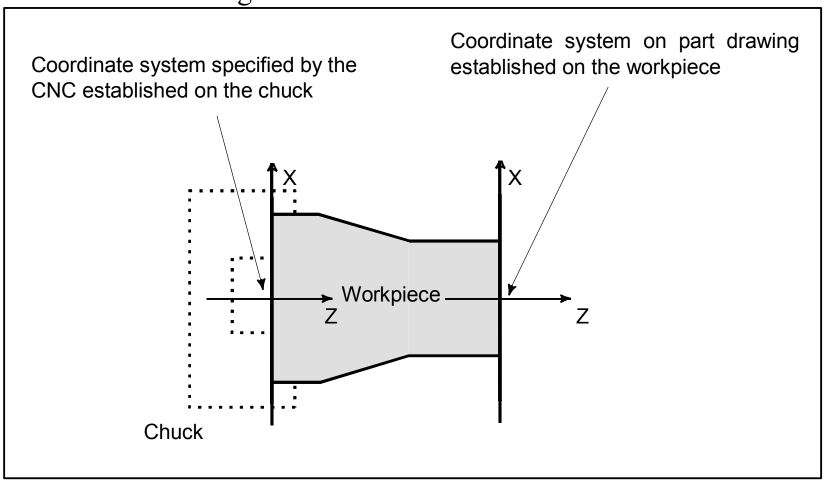


Fig. 1.3.2 (c) Coordinate system specified by CNC and coordinate system on part drawing

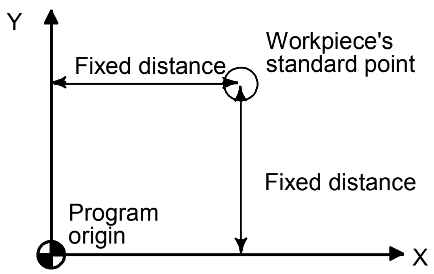
The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.
Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

- Methods of setting the two coordinate systems in the same position

M

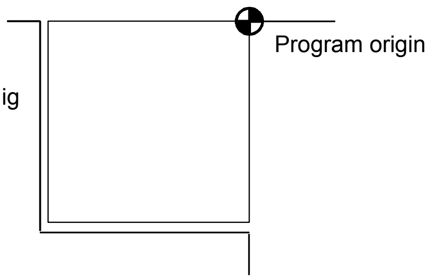
To set the two coordinate systems at the same position, simple methods shall be used according to workpiece shape, the number of machinings.

1. Using a standard plane and point of the workpiece.



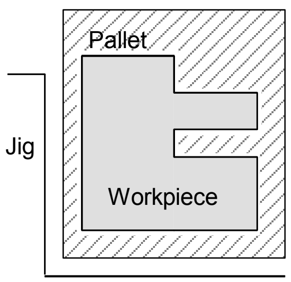
Bring the tool center to the workpiece standard point.
And set the coordinate system specified by CNC at this position.

2. Mounting a workpiece directly against the jig



Meet the tool center to the reference position. And set the coordinate specified by CNC at this position. (Jig shall be mounted on the point from the reference

3. Mounting a workpiece on a pallet, then mounting the workpiece and pallet on the jig

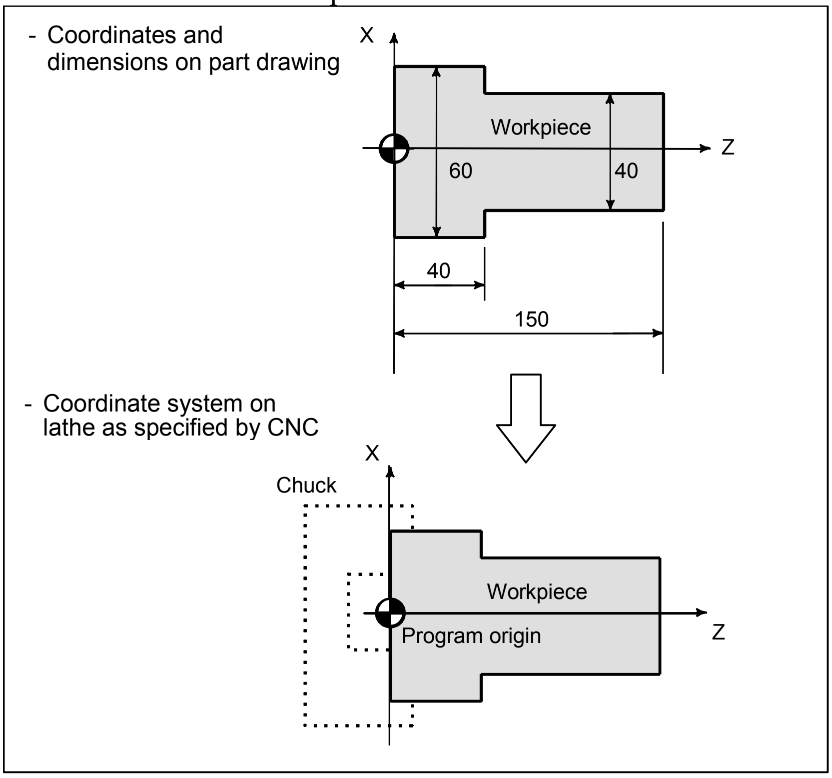


(Jig and coordinate system shall be specified by the same as (2)).

T

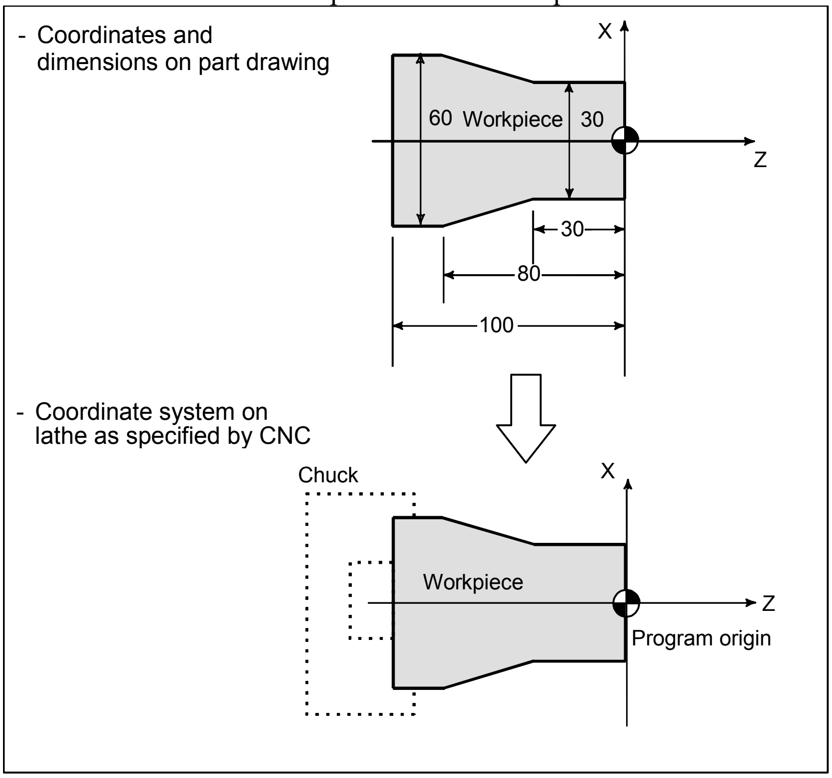
The following method is usually used to define two coordinate systems at the same location.

1 When coordinate zero point is set at chuck face



When the coordinate system on the part drawing and the coordinate system specified by the CNC are set at the same position, the program origin can be set on the chuck face.

2. When coordinate zero point is set at workpiece end face.



When the coordinate system on the part drawing and the coordinate system specified by the CNC are set at the same position, the program origin can be set on the end face of the workpiece.

1.3.3 How to Indicate Command Dimensions for Moving the Tool (Absolute, Incremental Commands)

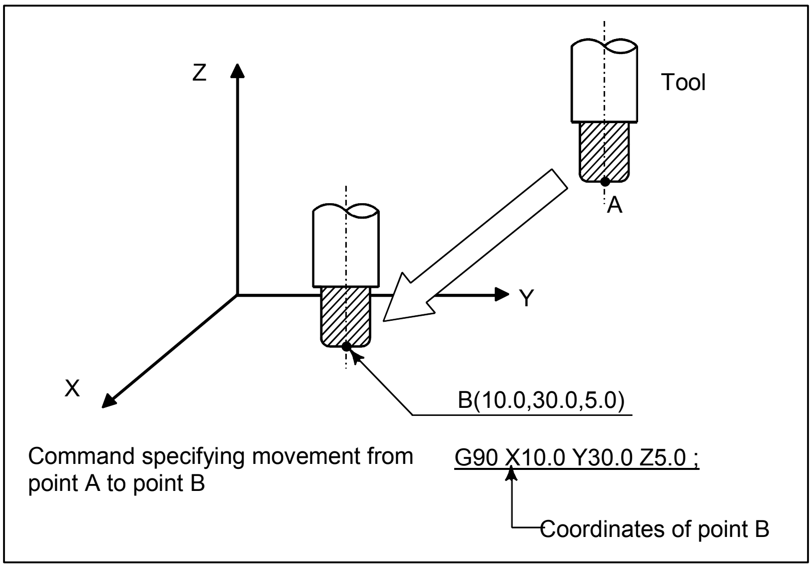
Explanation

Command for moving the tool can be indicated by absolute command or incremental command (See II-8.1).

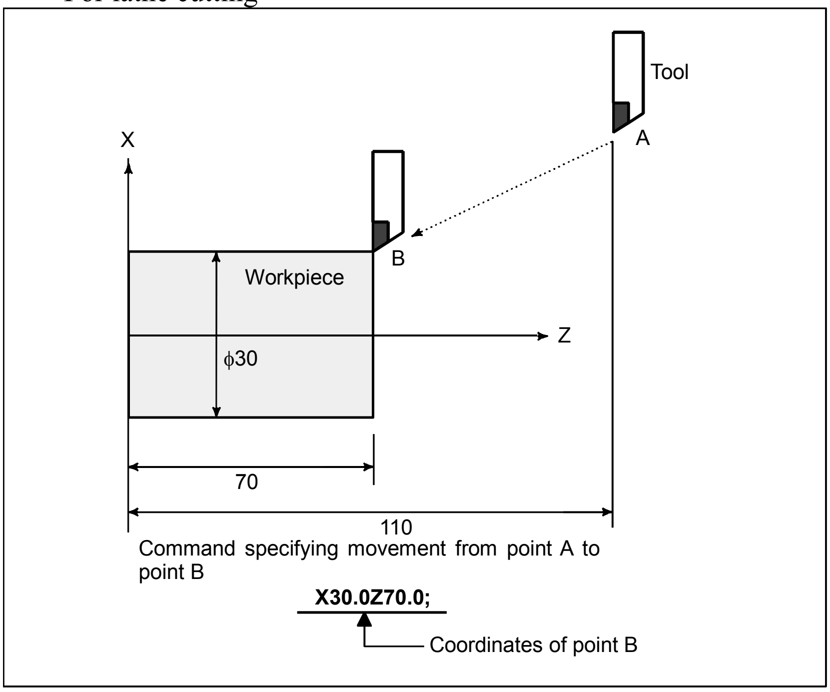
- Absolute command

The tool moves to a point at "the distance from zero point of the coordinate system" that is to the position of the coordinate values.

- For milling machining



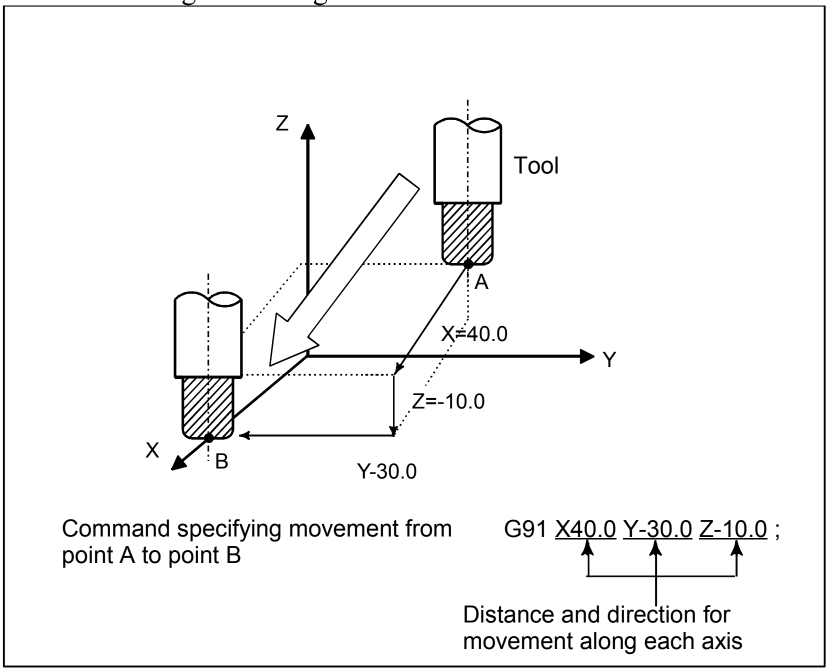
- For lathe cutting



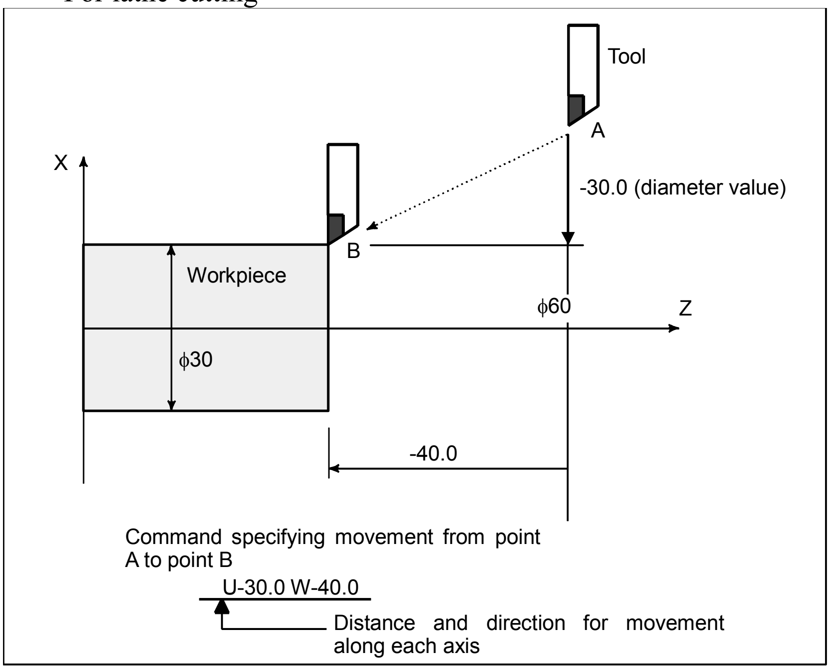
- Incremental command

Specify the distance from the previous tool position to the next tool position.

- For milling machining



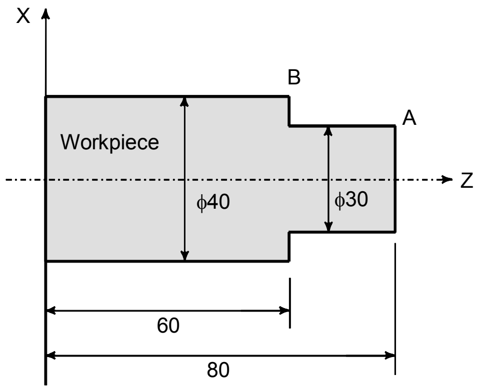
- For lathe cutting



- Diameter programming / radius programming

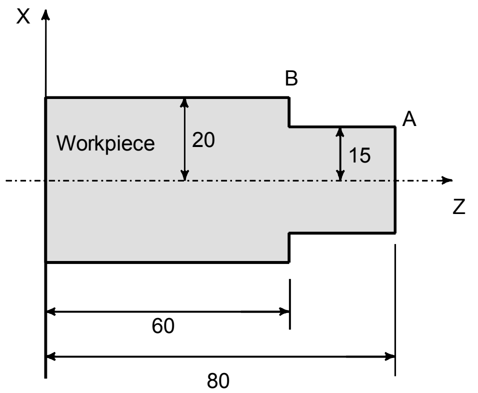
Dimensions of the X axis can be set in diameter or in radius. Diameter programming or radius programming is employed independently in each machine.

1. Diameter programming
   In diameter programming, specify the diameter value indicated on the drawing as the value of the X axis.



Coordinate values of points A and B **A(30.0, 80.0), B(40.0, 60.0)**

2. Radius programming
   In radius programming, specify the distance from the center of the workpiece, i.e. the radius value as the value of the X axis.

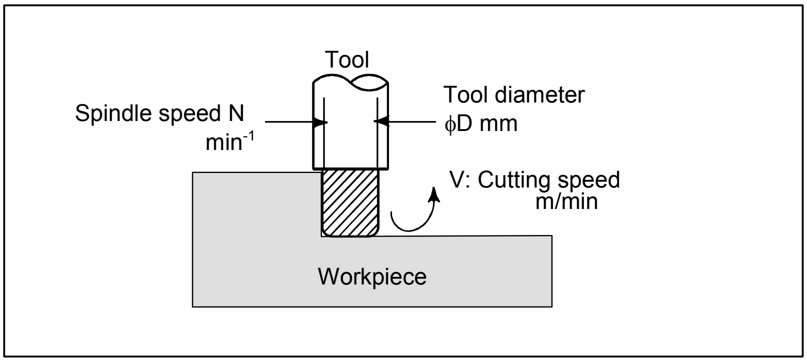


Coordinate values of points A and B **A(15.0, 80.0), B(20.0, 60.0)**

1.4 CUTTING SPEED - SPINDLE FUNCTION

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed.
As for the CNC, the cutting speed can be specified by the spindle speed in min^{-1} unit.

- For milling machining

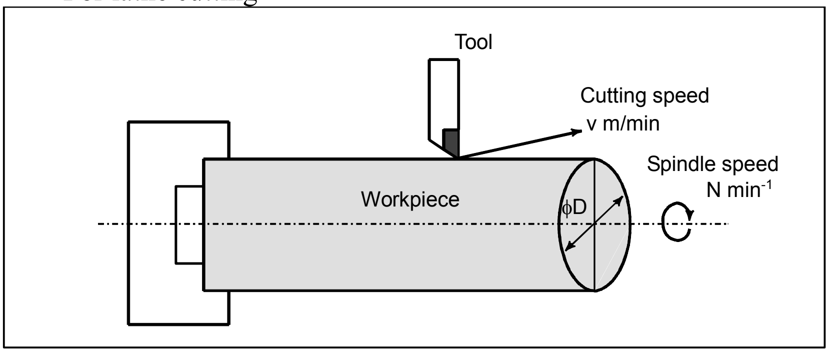


<When a workpiece should be machined with a tool 100 mm in diameter at a cutting speed of 80 m/min.>
The spindle speed is approximately 250 min^{-1}, which is obtained from $N=1000v/\pi D$. Hence the following command is required:

S250;

Commands related to the spindle speed are called the spindle speed function (See II-9) .

- For lathe cutting



<When a workpiece 200 mm in diameter should be machined at a cutting speed of 300 m/min.>
The spindle speed is approximately 478 min^{-1}, which is obtained from $N=1000v/\pi D$. Hence the following command is required:

S478 ;

Commands related to the spindle speed are called the spindle speed function (See II-9).
The cutting speed v (m/min) can also be specified directly by the speed value. Even when the workpiece diameter is changed, the CNC changes the spindle speed so that the cutting speed remains constant.
This function is called the constant surface speed control function (See II-9.3).

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION

Overview

For each of various types of machining (such as drilling, tapping, boring, and milling for milling machining, or rough machining, semifinish machining, finish machining, threading, and grooving for lathe cutting), a necessary tool is to be selected. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.

Examples

M

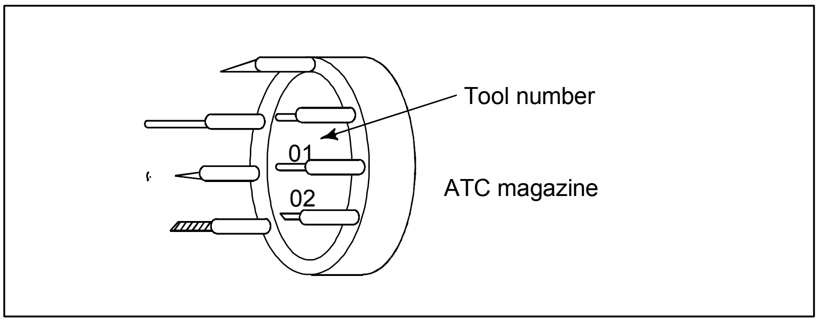


Fig. 1.5 (a) Tool used for various machining

<When No.01 is assigned to a drilling tool>
When the tool is stored at location 01 in the ATC magazine, the tool can be selected by specifying T01. This is called the tool function (See II-10).

T

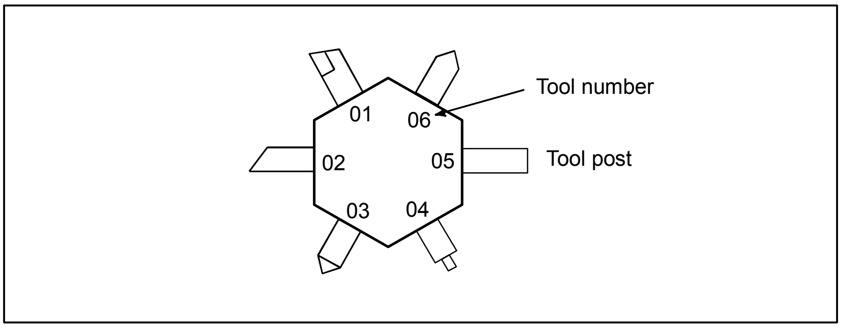


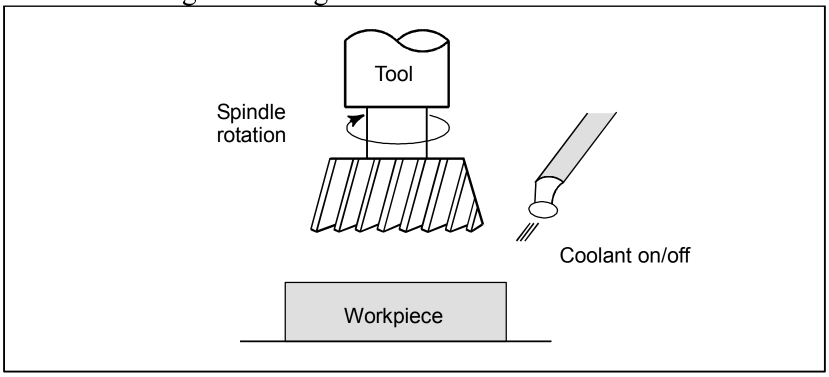
Fig. 1.5 (b) Tool used for various machining

<When No.01 is assigned to a roughing tool>
When the tool is stored at location 01 of the tool post, the tool can be selected by specifying T0101. This is called the tool function (See II-10).

1.6 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION

When a workpiece is actually machined with a tool, the spindle is rotated, coolant is supplied, and the chuck is opened/closed. So, control needs to be exercised on the spindle motor of the machine, coolant valve on/off operation, and chuck open/close operation.

- For milling machining



- For lathe cutting

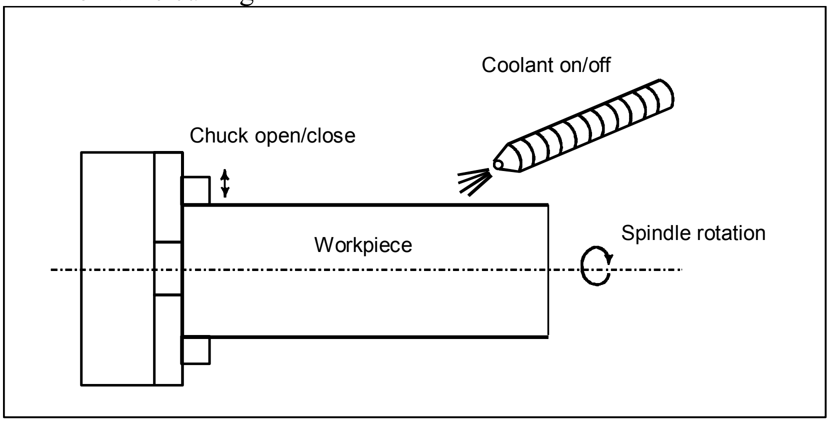


Fig. 1.6 (a) Auxiliary function

The function of specifying the on-off operations of the components of the machine is called the auxiliary function. In general, the function is specified by an M code (See II-11).
For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

1.7 PROGRAM CONFIGURATION

A group of commands given to the CNC for operating the machine is called the program. By specifying the commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off. In the program, specify the commands in the sequence of actual tool movements.

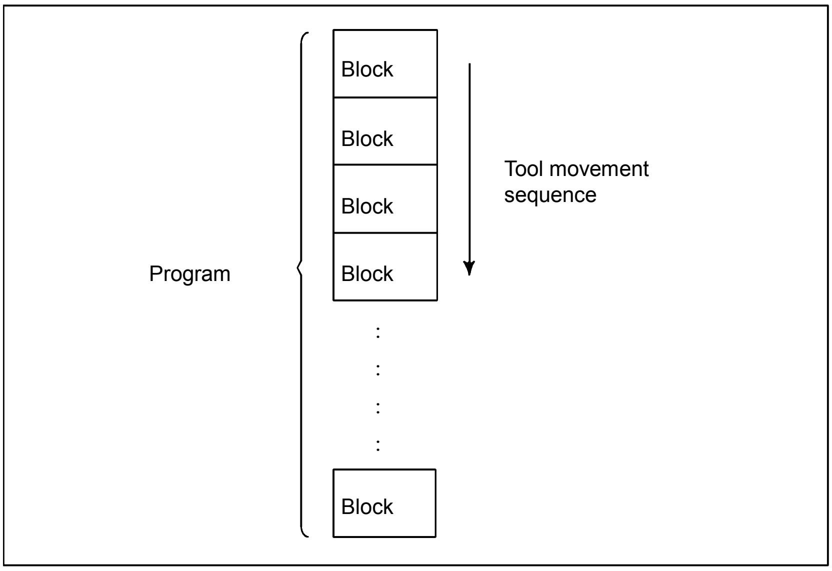


Fig. 1.7 (a) Program configuration

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number (See II-13).

Explanation

The block and the program have the following configurations.

- Block

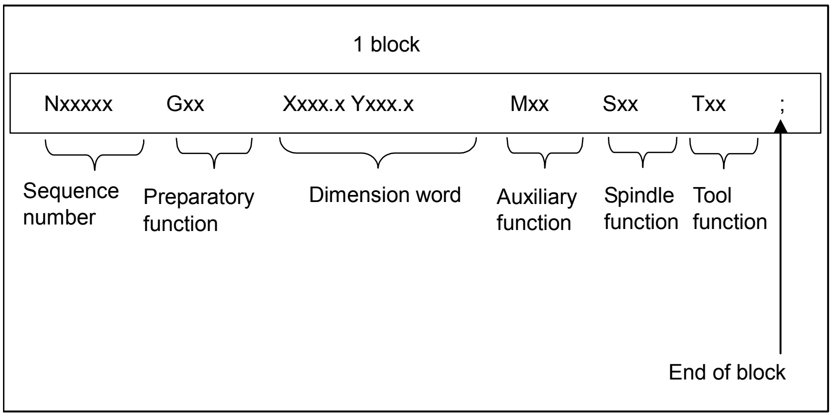


Fig. 1.7 (b) Block configuration

A block starts with a sequence number to identify the block and ends with an end-of-block code.
This manual indicates the end-of-block code by ; (LF in the ISO code and CR in the EIA code).
The contents of the dimension word depend on the preparatory function. In this manual, the portion of the dimension word may be represent as IP_.

- Program

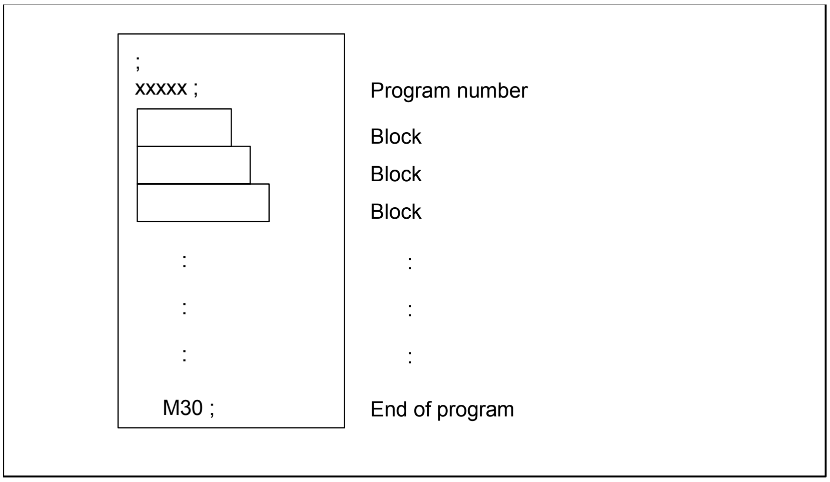


Fig. 1.7 (c) Program configuration

Normally, a program number is specified after the end-of-block (;) code at the beginning of the program, and a program end code (M02 or M30) is specified at the end of the program.

- Main program and subprogram

When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands of the subprogram are executed. When execution of the subprogram is finished, the sequence returns to the main program.

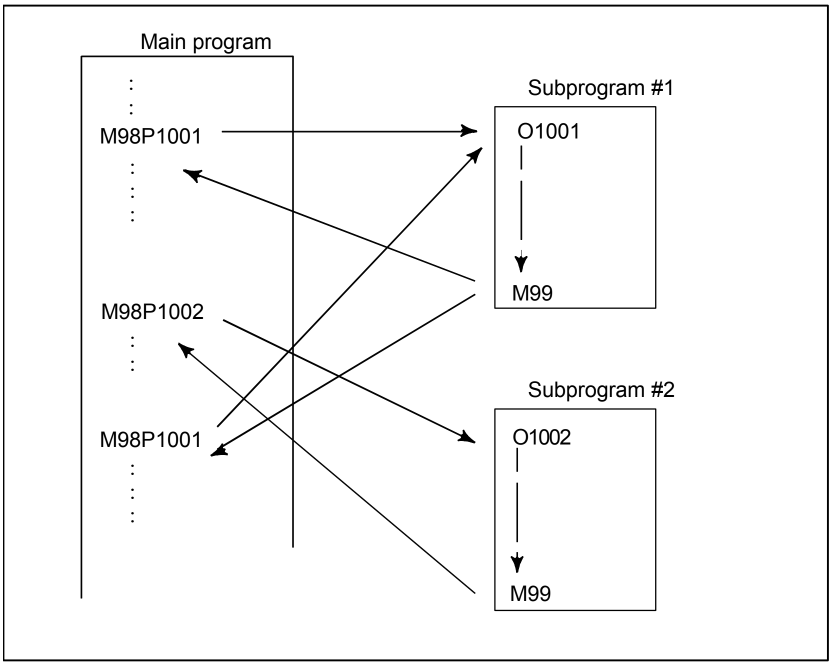
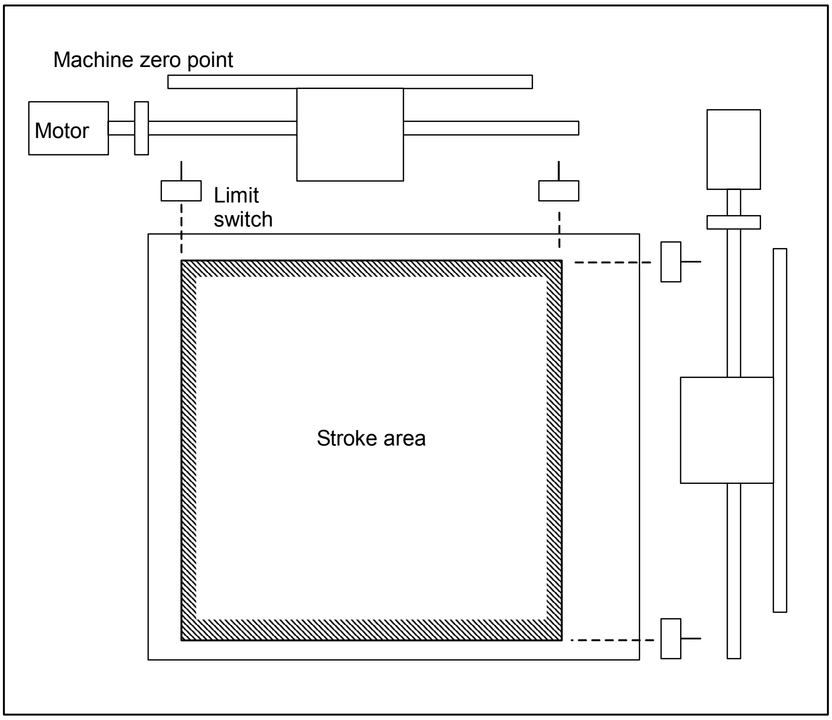


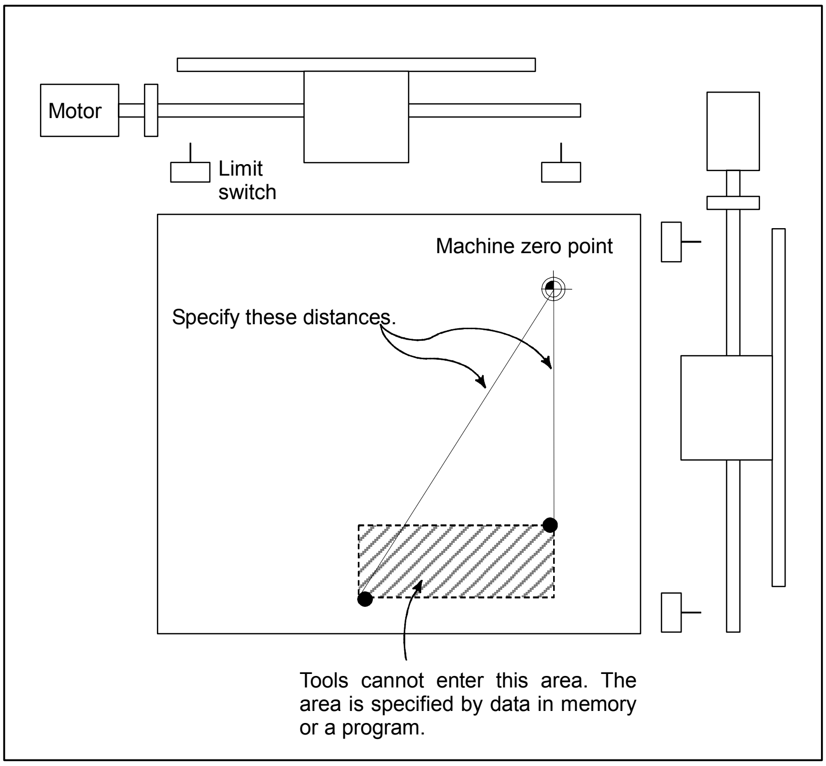
Fig. 1.7 (d) Subprogram execution

1.8 TOOL MOVEMENT RANGE - STROKE

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends. The range in which tools can move is called the stroke.



Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory. This function is called stroke check (see III-6.3).


Motor
Limit switch
Machine zero point
Specify these distances.
Tools cannot enter this area. The area is specified by data in memory or a program.

2 CONTROLLED AXES

2.1 NUMBER OF CONTROLLED AXES

Explanation

The number of controlled axes used with this NC system depends on the model and system control type as indicated below.

		Series 30*i*-A Series 300*i*-A Series 300*is*-A	**Series 31*i*-A5 Series 310*i*-A5 Series 310*is*-A5**	**Series 31*i*-A Series 310*i*-A Series 310*is*-A**	**Series 32*i*-A Series 320*i*-A Series 320*is*-A**
Number of basic controlled axes	Lathe system	2 axes	2 axes	2 axes	2 axes
	Machining center system	3 axes	3 axes	3 axes	3 axes
Controlled axes expansion (total) (including Cs axes and PMC axes)		Max. 32 axes	Max. 20 axes	Max. 20 axes	Max. 9 axes
Basic simultaneously controlled axes (each path)		2 axes	2 axes	2 axes	2 axes
Simultaneously controlled axes expansion (total / each path)		Max. 24 axes	Max. 12 axes	Max. 12 axes	Max. 5 axes

NOTE

1. The maximum number of controlled axes that can be used is limited depending on the option configuration. Refer to the manual provided by the machine tool builder for details.
2. The number of simultaneously controllable axes for manual operation (jog feed, manual reference position return, or manual rapid traverse) is 1 or 3 (1 when parameter JAX (No. 1002#0) is set to 0 and 3 when it is set to 1).

2.2 NAMES OF AXES

Explanation

The move axes of machine tools are assigned names. These names are referred to as addresses or axis names. Axis names are determined according to the machine tool. The naming rules comply with standards such as the ISO standards.
With complex machines, one character would become insufficient for representing axis names. So, up to three characters can be used for axis names. A move axis may be named "X", "X1", or "XA1". The first character of the three characters is called the first axis name character, the second character is called the second axis name character, and third character is called the third axis name character.
Example)

X A 1

3rd axis name character
2nd axis name character
1st axis name character

> **NOTE**
> 1 Axis names are predetermined according to the machine used. Refer to the manual supplied by the machine tool builder.
> 2 Since many ordinary machines use one character to represent each address, one-character addresses are used in the description in this manual.

2.3 INCREMENT SYSTEM

Explanation

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm, inches, or deg.
Five types of increment systems are available as indicated in Table 2.3 (a). For each axis, an increment system can be set using a bit from bit 0 to bit 3 (ISA, ISC, ISD, or ISE) of parameter No. 1013.
When IS-D or IS-E is to be selected, the corresponding option is required.

Table 2.3 (a) Increment system

Name of increment system	Least input increment		Least command increment	
IS-A	0.01	mm	0.01	mm
	0.001	inch	0.001	inch
	0.01	deg	0.01	deg
IS-B	0.001	mm	0.001	mm
	0.0001	inch	0.0001	inch
	0.001	deg	0.001	deg
IS-C	0.0001	mm	0.0001	mm
	0.00001	inch	0.00001	inch
	0.0001	deg	0.0001	deg
IS-D	0.00001	mm	0.00001	mm
	0.000001	inch	0.000001	inch
	0.00001	deg	0.00001	deg
IS-E	0.000001	mm	0.000001	mm
	0.0000001	inch	0.0000001	inch
	0.000001	deg	0.000001	deg

The least command increment is either metric or inch depending on the machine tool. Set metric or inch to the parameter INM (No.0100#0).
For selection between metric and inch for the least input increment, G code (G20 or G21) or a setting parameter selects it.
Combined use of the inch system and the metric system is not allowed. There are functions that cannot be used between axes with different unit systems (circular interpolation, cutter compensation, etc.). For the increment system, see the machine tool builder's manual.

NOTE

1. The unit (mm or inch) in the table is used for indicating a diameter value for diameter programming (when bit 3 (DIA) of parameter No. 1006 is set to 1) or a radius value for radius programming.
2. Some increment systems are unavailable depending on the model. For details, refer to "Descriptions" (B-63942EN).

2.4 MAXIMUM STROKE

Explanation

The maximum stroke controlled by this CNC is shown in the table below:
Maximum stroke = Least command increment × 99999999
(999999999 for IS-D and IS-E)
Commands that exceed the maximum stroke are not permitted.

Table 2.4 (a) Maximum strokes

Name of increment system	Least input increment	Maximum stroke
IS-A	0.01 mm	±999999.99 mm
	0.001 inch	±99999.999 inch
	0.01 deg	±999999.99 deg
IS-B	0.001 mm	±99999.999 mm
	0.0001 inch	±9999.9999 inch
	0.001 deg	±99999.999 deg
IS-C	0.0001 mm	±9999.9999 mm
	0.00001 inch	±999.99999 inch
	0.0001 deg	±9999.9999 deg
IS-D	0.00001 mm	±9999.99999 mm
	0.000001 inch	±999.999999 inch
	0.00001 deg	±9999.99999 deg
IS-E	0.000001 mm	±999.999999 mm
	0.0000001 inch	±99.9999999 inch
	0.000001 deg	±999.999999 deg

NOTE
1 The actual stroke depends on the machine tool.
2 The unit (mm or inch) in the table is used for indicating a diameter value for diameter programming (when bit 3 (DIA) of parameter No. 1006 is set to 1) or a radius value for radius programming.
3 Some increment systems are unavailable depending on the model. For details, refer to "Descriptions" (B-63942EN).

3 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified.
Modal G code	The G code is effective until another G code of the same group is specified.

(Example)

G01 and G00 are modal G codes in group 01.

```
G01 X_ ;
    Z_ ; } G01 is effective in this range.
    X_ ;
G00 Z_ ; } G00 is effective in this range.
    X_ ;
G01 X_ ;
  :
```

T

There are three G code systems in the lathe system : A, B, and C (Table 3.1(a)). Select a G code system using the parameters GSB and GSC (No. 3401#6 and #7). To use G code system B or C, the corresponding option is needed. Generally, User's Manual describes the use of G code system A, except when the described item can use only G code system B or C. In such cases, the use of G code system B or C is described.

Explanation

1. When the clear state (parameter CLR (No. 3402#6)) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with ◤ as indicated in Table.
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by parameter G23 (No. 3402#7). However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting parameter G01 (No. 3402#0).
   (5) The user can select G90 or G91 by setting parameter G91 (No. 3402#3).
   When G code system B or C is used in the lathe system, setting parameter G91 (No. 3402#3) determines which code, either G90 or G91, is effective.
   (6) In the machining center system, the user can select G17, G18, or G19 by setting parameters G18 and G19 (No. 3402#1 and #2).
2. G codes other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010 occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a canned cycle for drilling, the canned cycle for drilling is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle for drilling.
6. G codes are indicated by group.
7. The group of G60 is switched according to the setting of the parameter MDL (No. 5431#0). (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

T

8. When G code system A is used, absolute or incremental programming is specified not by a G code (G90/G91) but by an address word (X/U, Z/W, C/H, Y/V). Only the initial level is provided at the return point of the canned cycle for drilling..

3.1 G CODE LIST IN THE MACHINING CENTER SYSTEM

M

Table 3.1 (a) G code list

G code	Group	Function	
G00	01	Positioning (rapid traverse)	
G01		Linear interpolation (cutting feed)	
G02		Circular interpolation CW or helical interpolation CW	
G03		Circular interpolation CCW or helical interpolation CCW	
G02.2, G03.2		Involute interpolation CW/CCW	
G02.3, G03.3		Exponential interpolation CW/CCW	
G02.4, G03.4		Three-dimensional coordinate conversion CW/CCW	
G04	00	Dwell	
G05		AI contour control (high-precision contour control compatible command)	
G05.1		AI contour control / Nano smoothing / Smooth interpolation	
G05.4		HRV3,4 on/off	
G06.2	01	NURBS interpolation	
G07	00	Hypothetical axis interpolation	
G07.1 (G107)		Cylindrical interpolation	
G08		AI contour control (advanced preview control compatible command)	
G09		Exact stop	
G10		Programmable data input	
G10.6		Tool retract and recover	
G10.9		Programmable switching of diameter/radius specification	
G11		Programmable data input mode cancel	
G12.1	21	Polar coordinate interpolation mode	
G13.1		Polar coordinate interpolation cancel mode	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20 (G70)	06	Input in inch	
G21 (G71)		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G25	19	Spindle speed fluctuation detection off	
G26		Spindle speed fluctuation detection on	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G30.1		Floating reference position return	
G31		Skip function	
G31.8		EGB-axis skip	
G33	01	Threading	
G34		Variable lead threading	
G35		Circular threading CW	
G36		Circular threading CCW	

Table 3.1 (a) G code list

G code	Group	Function
G37	00	Automatic tool length measurement
G38		Cutter or tool nose radius compensation : preserve vector
G39		Cutter or tool nose radius compensation : corner circular interpolation
G40	07	Cutter or tool nose radius compensation : cancel Three-dimensional cutter compensation : cancel
G41		Cutter or tool nose radius compensation : left Three-dimensional cutter compensation : left
G41.2		Cutter compensation for 5-axis machining : left (type 1)
G41.3		Cutter compensation for 5-axis machining : (leading edge offset)
G41.4		Cutter compensation for 5-axis machining : left (type 1) (FS16i-compatible command)
G41.5		Cutter compensation for 5-axis machining : left (type 1) (FS16i-compatible command)
G41.6		Cutter compensation for 5-axis machining : left (type 2)
G42		Cutter or tool nose radius compensation : right Three-dimensional cutter compensation : right
G42.2		Cutter compensation for 5-axis machining : right (type 1)
G42.4		Cutter compensation for 5-axis machining : right (type 1) (FS16i-compatible command)
G42.5		Cutter compensation for 5-axis machining : right (type 1) (FS16i-compatible command)
G42.6		Cutter compensation for 5-axis machining : right (type 2)
G40.1	19	Normal direction control cancel mode
G41.1		Normal direction control on : left
G42.1		Normal direction control on : right
G43	08	Tool length compensation +
G44		Tool length compensation -
G43.1	08	Tool length compensation in tool axis direction
G43.4		Tool center point control (type 1)
G43.5		Tool center point control (type 2)
G45	00	Tool offset increase
G46		Tool offset decrease
G47		Tool offset double increase
G48		Tool offset double decrease
G49 (G49.1)	08	Tool length compensation cancel
G50	11	Scaling cancel
G51		Scaling
G50.1	22	Programmable mirror image cancel
G51.1		Programmable mirror image
G50.2	31	Polygon turning cancel
G51.2		Polygon turning
G52	00	Local coordinate system setting
G53		Machine coordinate system setting
G53.1		Tool axis direction control
G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning

Table 3.1 (a) G code list

G code	Group	Function
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call A
G66.1		Macro modal call B
G67		Macro modal call A/B cancel
G68	16	Coordinate system rotation start or 3-dimensional coordinate conversion mode on
G69		Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off
G68.2		Feature coordinate system selection
G72.1	00	Figure copy (rotation copy)
G72.2		Figure copy (linear copy)
G73	09	Peck drilling cycle
G74		Left-handed tapping cycle
G76		Fine boring cycle
G80		Canned cycle cancel
G80.5	24	Electronic gear box 2 pair: synchronization cancellation
G80.8	34	Electronic gear box: synchronization cancellation
G81	09	Drilling cycle or spot boring cycle
G81.1	00	Chopping
G81.5	24	Electronic gear box 2 pair: synchronization start
G81.8	34	Electronic gear box: synchronization start
G82	09	Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS15 format)
G84.3		Left-handed rigid tapping cycle (FS15 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute programming
G91		Incremental programming
G91.1	00	Checking the maximum incremental amount specified
G92		Setting for workpiece coordinate system or clamp at maximum spindle speed
G92.1		Workpiece coordinate system preset
G93	05	Inverse time feed
G94		Feed per minute
G95		Feed per revolution
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
G107	00	Cylindrical interpolation
G112	21	Polar coordinate interpolation mode
G113		Polar coordinate interpolation mode cancel

3.2 G CODE LIST IN THE LATHE SYSTEM

T

Table 3.2 (a) G code list

G code system			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or helical interpolation CW
G03	G03	G03		Circular interpolation CCW or helical interpolation CCW
G02.2	G02.2	G02.2		Involute interpolation CW
G02.3	G02.3	G02.3		Exponential interpolation CW
G02.4	G02.4	G02.4		Three-dimensional coordinate conversion CW
G03.2	G03.2	G03.2		Involute interpolation CCW
G03.3	G03.3	G03.3		Exponential interpolation CCW
G03.4	G03.4	G03.4		Three-dimensional coordinate conversion CCW
G04	G04	G04	00	Dwell
G05	G05	G05		AI contour control (command compatible with high precision contour control)
G05.1	G05.1	G05.1		AI contour control / Nano smoothing / Smooth interpolation
G05.4	G05.4	G05.4		HRV3,4 on/off
G06.2	G06.2	G06.2	01	NURBS interpolation
G07	G07	G07	00	Hypothetical axis interpolation
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G08	G08	G08		Advanced preview control
G09	G09	G09		Exact stop
G10	G10	G10		Programmable data input
G10.6	G10.6	G10.6		Tool retract and recover
G10.9	G10.9	G10.9		Programmable switching of diameter/radius specification
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G15	G15	G15	24	Polar coordinate command cancel
G16	G16	G16		Polar coordinate command
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on

Table 3.2 (a) G code list

G code system			Group	Function
A	B	C		
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G29	G29	G29		Movement from reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G30.1	G30.1	G30.1		Floating reference point return
G31	G31	G31		Skip function
G31.8	G31.8	G31.8		EGB-axis skip
G32	G33	G33	01	Threading
G34	G34	G34		Variable lead threading
G35	G35	G35		Circular threading CW
G36	G36	G36		Circular threading CCW (When the parameter G36 (No. 3405#3) is set to 1) or Automatic tool offset (X axis) (When the parameter G36 (No. 3405#3) is set to 0)
G37	G37	G37		Automatic tool offset (Z axis) (When the parameter G36 (No. 3405#3) is set to 0)
G37.1	G37.1	G37.1		Automatic tool offset (X axis) (When the parameter G36 (No. 3405#3) is set to 1)
G37.2	G37.2	G37.2		Automatic tool offset (Z axis) (When the parameter G36 (No. 3405#3) is set to 1)
G38	G38	G38		Cutter compensation of tool nose radius compensation: with vector held
G39	G39	G39		Cutter compensation of tool nose radius compensation: corner rounding interpolation
G40	G40	G40	07	Cutter compensation of tool nose radius compensation : cancel
G41	G41	G41		Cutter compensation of tool nose radius compensation : left
G42	G42	G42		Cutter compensation of tool nose radius compensation : right
G41.2	G41.2	G41.2		Cutter compensation for 5-axis machining : left (type 1)
G41.3	G41.3	G41.3		Cutter compensation for 5-axis machining : (leading edge offset)
G41.4	G41.4	G41.4		Cutter compensation for 5-axis machining : left (type 1) (FS16i-compatible command)
G41.5	G41.5	G41.5		Cutter compensation for 5-axis machining : left (type 1) (FS16i-compatible command)
G41.6	G41.6	G41.6		Cutter compensation for 5-axis machining : left (type 2)
G42.2	G42.2	G42.2		Cutter compensation for 5-axis machining : right (type 1)
G42.4	G42.4	G42.4		Cutter compensation for 5-axis machining : right (type 1) (FS16i-compatible command)
G42.5	G42.5	G42.5		Cutter compensation for 5-axis machining : right (type 1) (FS16i-compatible command)
G42.6	G42.6	G42.6		Cutter compensation for 5-axis machining : right (type 2)
G43	G43	G43	23	Tool length compensation +
G44	G44	G44		Tool length compensation -
G43.1	G43.1	G43.1		Tool length compensation in tool axis direction
G43.4	G43.4	G43.4		Tool center point control (type 1)
G43.5	G43.5	G43.5		Tool center point control (type 2)
G43.7 (G44.7)	G43.7 (G44.7)	G43.7 (G44.7)		Tool offset (lathe system ATC type)
G49 (G49.1)	G49 (G49.1)	G49 (G49.1)		Tool length compensation cancel

Table 3.2 (a) G code list

G code system A	G code system B	G code system C	Group	Function
G50	G92	G92	00	Coordinate system setting or max. spindle speed clamp
G50.3	G92.1	G92.1		Workpiece coordinate system preset
-	G50	G50	18	Scaling cancel
-	G51	G51		Scaling
G50.1	G50.1	G50.1	22	Programmable mirror image cancel
G51.1	G51.1	G51.1		Programmable mirror image
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygon turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygon turning
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G53.1	G53.1	G53.1		Tool axis direction control
G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G60	G60	G60	00	Single direction positioning
G61	G61	G61	15	Exact stop mode
G62	G62	G62		Automatic corner override mode
G63	G63	G63		Tapping mode
G64	G64	G64		Cutting mode
G65	G65	G65	00	Macro call
G66	G66	G66	12	Macro modal call A
G66.1	G66.1	G66.1		Macro modal call B
G67	G67	G67		Macro modal call A/B cancel
G68	G68	G68	04	Mirror image on for double turret or balance cutting mode
G68.1	G68.1	G68.1	17	Coordinate system rotation start or 3-dimensional coordinate conversion mode on
G68.2	G68.2	G68.2		Feature coordinate system selection
G69	G69	G69	04	Mirror image off for double turret or balance cutting mode cancel
G69.1	G69.1	G69.1	17	Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating cycle
G74	G74	G76		End face peck drilling cycle
G75	G75	G77		Outer diameter/internal diameter drilling cycle
G76	G76	G78		Multiple-thread cutting cycle
G72.1	G72.1	G72.1		Figure copy (rotation copy)
G72.2	G72.2	G72.2		Figure copy (parallel copy)

Table 3.2 (a) G code list

G code system			Group	Function
A	B	C		
G80	G80	G80	10	Canned cycle cancel for drilling
G80.5	G80.5	G80.5	27	Electronic gear box 2 pair: synchronization cancellation
G80.8	G80.8	G80.8	28	Electronic gear box: synchronization cancellation
G81	G81	G81	10	Spot drilling (FS15-T format)
G81.5	G81.5	G81.5	27	Electronic gear box 2 pair: synchronization start
G81.8	G81.81	G81.8	28	Electronic gear box: synchronization start
G82	G82	G82	10	Counter boring (FS15-T format)
G83	G83	G83		Cycle for face drilling
G83.1	G83.1	G83.1		High-speed peck drilling cycle (FS15-T format)
G83.5	G83.5	G83.5		High-speed peck drilling cycle
G83.6	G83.6	G83.6		Peck drilling cycle
G84	G84	G84		Cycle for face tapping
G84.2	G84.2	G84.2		Rigid tapping cycle (FS15-T format)
G85	G85	G85		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G87.5	G87.5	G87.5		High-speed peck drilling cycle
G87.6	G87.6	G87.6		Peck drilling cycle
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Threading cycle
G94	G79	G24		End face turning cycle
G91.1	G91.1	G91.1	00	Maximum specified incremental amount check
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G93	G93	G93	05	Inverse time feed
G98	G94	G94		Feed per minute
G99	G95	G95		Feed per revolution
-	G90	G90	03	Absolute programming
-	G91	G91		Incremental programming
-	G98	G98	11	Canned cycle : return to initial level
-	G99	G99		Canned cycle : return to R point level

4 INTERPOLATION FUNCTIONS

Interpolation functions specify the way to make an axis movement (in other words, a movement of the tool with respect to the workpiece or table).

4.1 POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.
In the absolute command, coordinate value of the end point is programmed.
In the incremental command the distance the tool moves is programmed.

Format

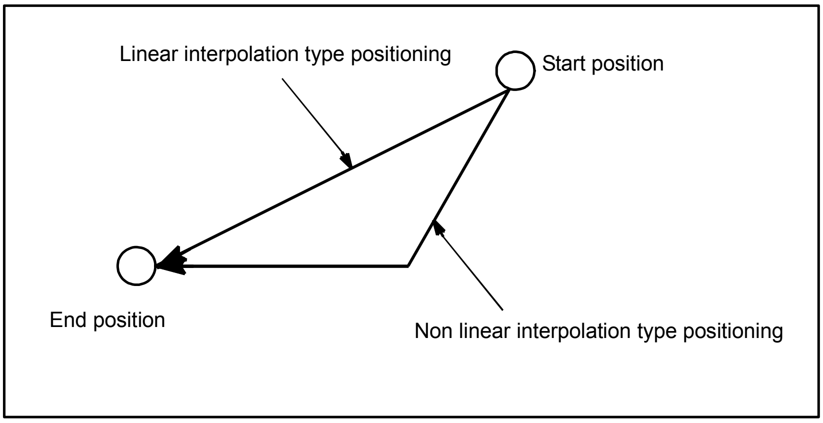
G00 IP_ ;

IP_ : For an absolute command, the coordinates of an end point, and for an incremental command, the distance the tool moves.

Explanation

Either of the following tool paths can be selected according to bit 1 (LRP) of parameter No. 1401.

- Nonlinear interpolation type positioning
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.
- Linear interpolation type positioning. The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



The rapid traverse rate in G00 command is set to the parameter No. 1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position.
"In-position " means that the feed motor is within the specified range.
This range is determined by the machine tool builder by setting to parameter (No. 1826).

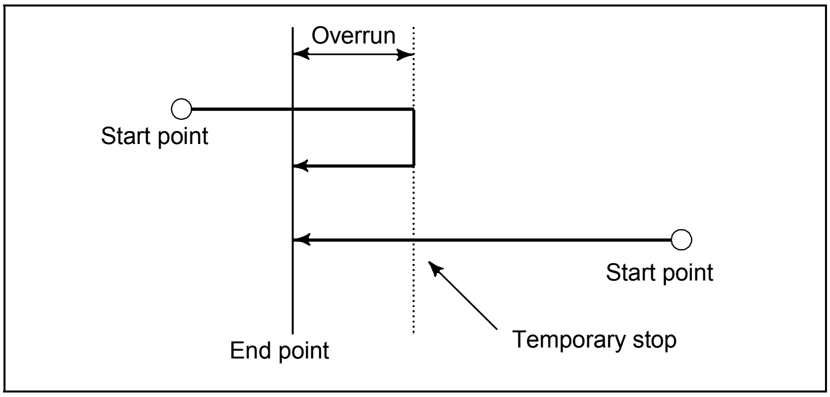
Limitation

The rapid traverse rate cannot be specified in the address F.
Even if linear interpolation type positioning is specified, nonlinear type interpolation positioning is used in the following cases. Therefore, be careful to ensure that the tool does not foul the workpiece.

- G28 specifying positioning between the reference and intermediate positions.
- G53

4.2 SINGLE DIRECTION POSITIONING (G60)

For accurate positioning without play of the machine (backlash), final positioning from one direction is available.



Format

G60 IP_ ;

IP_ : For an absolute command, the coordinates of an end point, and for an incremental command, the distance the tool moves.

Explanation

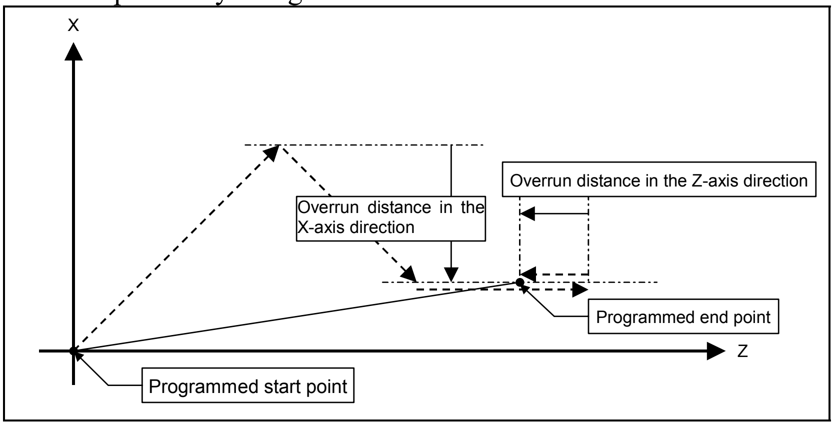
An overrun and a positioning direction are set by the parameter (No. 5440). Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.
G60, which is an one-shot G-code, can be used as a modal G-code in group 01 by setting 1 to the parameter MDL (No. 5431#0).
This setting can eliminate specifying a G60 command for every block. Other specifications are the same as those for an one-shot G60 command. When an one-shot G code is specified in the single direction positioning mode, the one-shot G command is effective like G codes in group 01.

(Example)

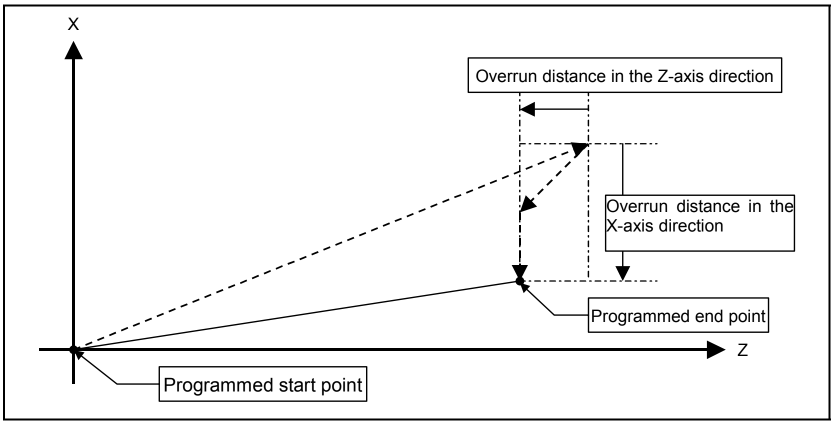
When one-shot G60 commands are used.	
G90;	
G60 X0Y0;	Single direction positioning
G60 X100;	Single direction positioning
G60 Y100;	Single direction positioning
G04 X10;	
G00 X0Y0;	
When modal G60 command is used.	
G90G60;	Single direction positioning mode start
X0Y0;	Single direction positioning
X100;	Single direction positioning
Y100;	Single direction positioning
G04X10;	
G00X0 Y0;	Single direction positioning mode cancel

- Overview of operation

- **In the case of positioning of non-linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 0)**
  As shown below, single direction positioning is performed independently along each axis.



- **In the case of positioning of linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 1)**
  Positioning of interpolation type is performed until the tool once stops before or after a specified end point. Then, the tool is positioned independently along each axis until the end point is reached.



Limitation

- Single direction positioning is not performed along an axis for which no overrun distance is set in parameter No. 5440.
- Single direction positioning is not performed along an axis for which travel distance 0 is specified.
- The mirror image function is not applied in a parameter-set direction. Even in the mirror image mode, the direction of single direction positioning remains unchanged. If positioning of linear interpolation type is used, and the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, an alarm is issued. When switching mirror image in the middle of a program, disable looking ahead by specifying a

non-buffering M code. Then, switch mirror image when there is no look-ahead block.

- In the cylindrical interpolation mode (G07.1), single direction positioning cannot be used.
- In the polar coordinate interpolation mode (G12.1), single direction positioning cannot be used.
- When specifying single direction positioning on a machine that uses angular axis control, first position the angular axis then specify the positioning of the Cartesian axis. If the reverse specification order is used, or the angular axis and Cartesian axis are specified in the same block, an incorrect positioning direction can result.
- In positioning at a restart position by program restart function, single direction positioning is not performed.

M

- During canned cycle for drilling, no single direction positioning is effected in drilling axis.
- The single direction positioning does not apply to the shift motion in the canned cycles of G76 and G87.

T

- The G-code for single direction positioning is always G60, if G-code system is A or B or C in all case.
- The single direction positioning can not be commanded during the multiple repetitive cycle (G70-G76).
- No single direction positioning is effected in the drilling or patting axis, during canned cycle for drilling (G83-G89) and the rigid tapping (G84, G88). However, it can be commanded for positioning.
- The single direction positioning can not be commanded during the canned cycle (G90, G92, G94).
- During the single direction positioning mode (G60), the following G-code can not be commanded.
  G07.1, G12.1, G70-G76, G90-G94.

4.3 LINEAR INTERPOLATION (G01)

Tools can move along a line.

Format

G01 IP_ **F_ ;**

IP_ : For an absolute command, the coordinates of an end point, and for an incremental command, the distance the tool moves.

F_ : Speed of tool feed (Feedrate)

Explanation

A tools move along a line to the specified position at the feedrate specified in F.
The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.
The feedrate commanded by the F code is measured along the tool path. If the F code is not commanded, the feedrate is regarded as zero.
The feedrate of each axis direction is as follows.

G01 $\alpha\underline{\alpha}$ $\beta\underline{\beta}$ $\gamma\underline{\gamma}$ $\zeta\underline{\zeta}$ **Ff ;**

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \times f$

Feed rate of β axis direction : $F\beta = \frac{\beta}{L} \times f$

Feed rate of γ axis direction : $F\gamma = \frac{\gamma}{L} \times f$

Feed rate of ζ axis direction : $F\zeta = \frac{\zeta}{L} \times f$

$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \zeta^2}$

The feedrate of the rotary axis is commanded in the unit of deg/min (the unit is decimal point position).
When the straight line axis α (such as X, Y, or Z) and the rotating axis b (such as A, B, or C) are linearly interpolated, the feedrate is that in which the tangential feedrate in the α and β cartesian coordinate system is commanded by F(mm/min).
β-axis feedrate is obtained ; at first, the time required for distribution is calculated by using the above formula, then the β-axis feedrate unit is changed to deg/min.

A calculation example is as follows.
G91 G01 X20.0B40.0 F300.0 ;
This changes the unit of the C axis from 40.0 deg to 40mm with metric input. The time required for distribution is calculated as follows:

$$\frac{\sqrt{20^2+40^2}}{300} \fallingdotseq 0.14907(mm)$$

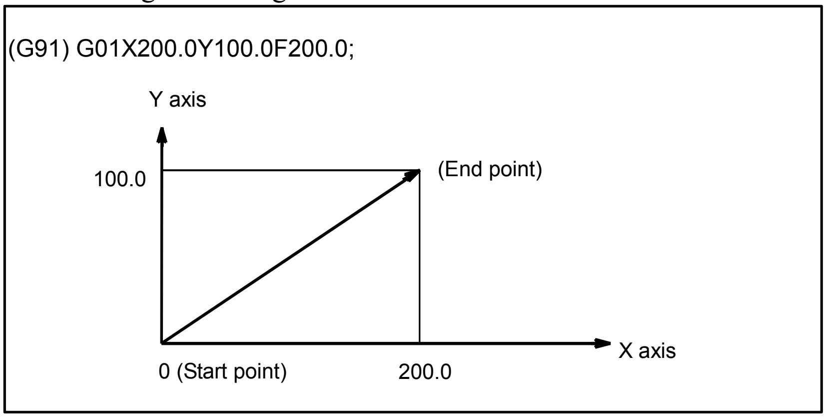
The feedrate for the C axis is

$$\frac{40}{0.14907} \fallingdotseq 268.3\,\deg/\min$$

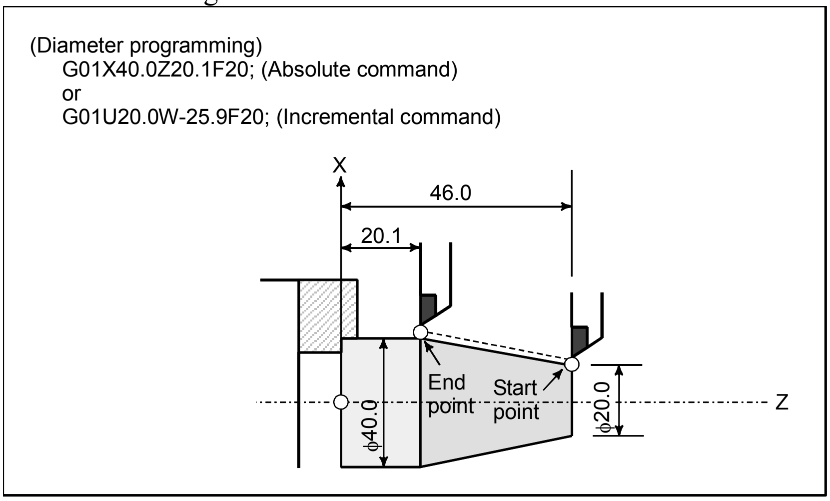
In simultaneous 3 axes control, the feedrate is calculated the same way as in 2 axes control.

Example
- Linear interpolation

• For milling machining

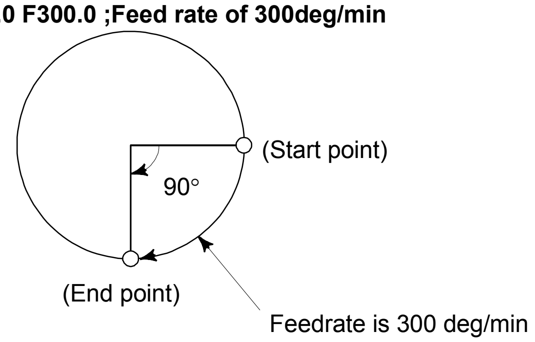


• For lathe cutting



- Feedrate for the rotary axis

G91G01C-90.0 F300.0 ;Feed rate of 300deg/min



4.4 CIRCULAR INTERPOLATION (G02, G03)

The command below will move a tool along a circular arc.

Format

Arc in the XpYp plane

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}Xp\_\ Yp\_\begin{Bmatrix}I\_\ J\_\\R\_\end{Bmatrix}F\_;$$

Arc in the ZpXp plane

$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}Zp\_\ Xp\_\begin{Bmatrix}I\_\ K\_\\R\_\end{Bmatrix}F\_;$$

Arc in the YpZp plane

$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Yp\_\ Zp\_\begin{Bmatrix}J\_\ K\_\\R\_\end{Bmatrix}F\_;$$

Command	Description
G17	Specification of arc on XpYp plane
G18	Specification of arc on ZpXp plane
G19	Specification of arc on YpZp plane
G02	Circular Interpolation Clockwise direction (CW)
G03	Circular Interpolation Counterclockwise direction (CCW)
Xp_	Command values of X axis or its parallel axis (set by parameter No. 1022)
Yp_	Command values of Y axis or its parallel axis (set by parameter No. 1022)
Zp_	Command values of Z axis or its parallel axis (set by parameter No. 1022)
I_	Xp axis distance from the start point to the center of an arc with sign
J_	Yp axis distance from the start point to the center of an arc with sign
K_	Zp axis distance from the start point to the center of an arc with sign
R_	Arc radius (with sign, radius value for lathe cutting)
F_	Feedrate along the arc

T

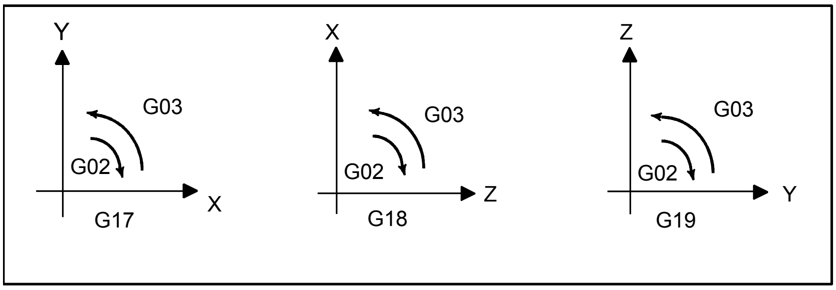
NOTE

The U-, V-, and W-axes can be used with G-codes B and C.

Explanation

- Direction of the circular interpolation

"Clockwise"(G02) and "counterclockwise"(G03) on the XpYp plane (ZpXp plane or YpZp plane) are defined when the XpYp plane is viewed in the positive-to-negative direction of the Zp axis (Yp axis or Xp axis, respectively) in the Cartesian coordinate system. See the figure below.



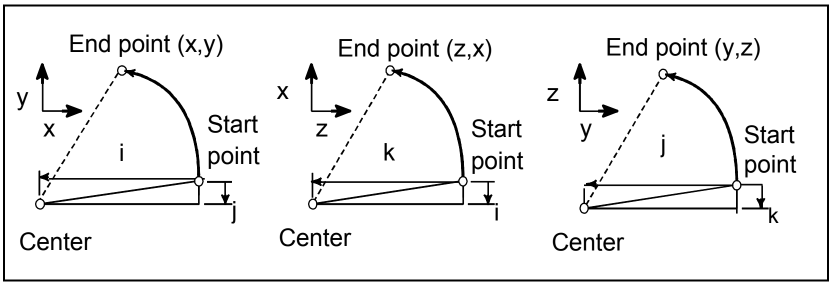
- Distance moved on an arc

The end point of an arc is specified by address Xp, Yp or Zp, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified with sign.

- Distance from the start point to the center of arc

The arc center is specified by addresses I, J, and K for the Xp, Yp, and Zp axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.

I, J, and K must be signed according to the direction.



I0,J0, and K0 can be omitted.
If the difference between the radius at the start point and that at the end point exceeds the permitted value in a parameter (No.3410), an alarm PS0020 occurs.

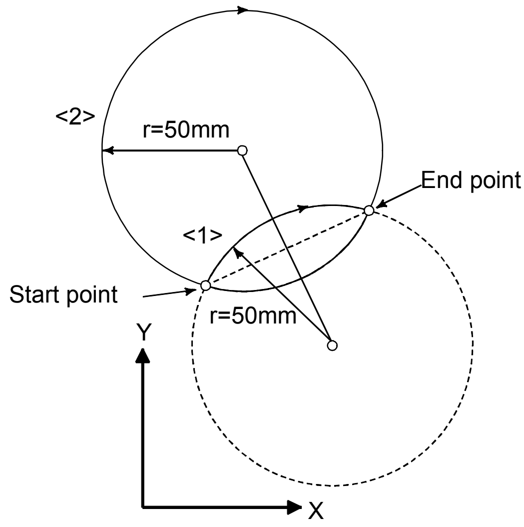
- Command for a circle

When Xp, Yp , and Zp are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.
G02 I_ ; Command for a circle

- Arc radius

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K.
In this case, one arc is less than 180°, and the other is more than 180° are considered. When an arc exceeding 180° is commanded, the radius must be specified with a negative value. If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0° is programmed
G02R_ ; (The cutter does not move.)

For arc <1> (less than 180°)
G91 G02 XP60.0 YP55.0 R50.0 F300.0 ;
For arc <2> (greater than 180°)
G91 G02 XP60.0 YP55.0 R-50.0 F300.0 ;



- Feedrate

The feedrate in circular interpolation is equal to the feedrate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feedrate is measured along the arc after the cutter compensation is applied

Limitation

- Simultaneously specifying R with I, J, and K

If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

- Specifying an axis that is not contained in the specified plane

If an axis not comprising the specified plane is commanded, an alarm PS0028 occurs.
For example,
For milling machining:
If the X-axis and a U-axis parallel to the X-axis are specified when the XY plane is specified
For lathe cutting:
If the X-axis and a U-axis parallel to the X-axis are specified when the ZX plane is specified with G code system B or C

- Specifying a semicircle with R

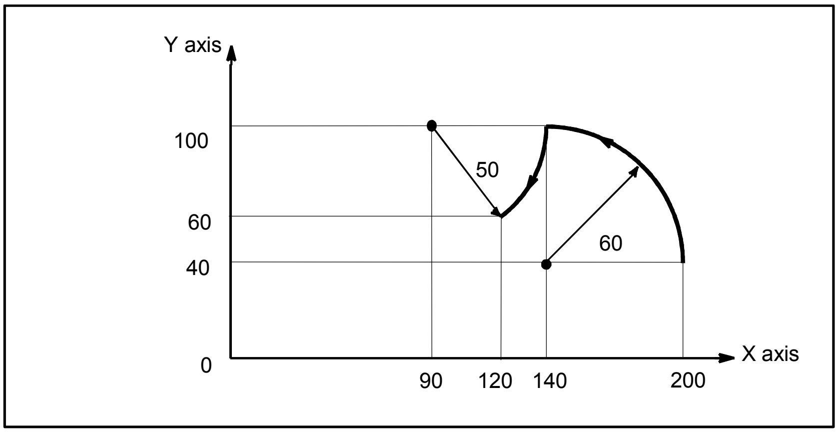
When an arc having a center angle approaching 180° is specified, the calculated center coordinates may contain an error. In such a case, specify the center of the arc with I, J, and K.

- Difference in the radius between the start and end points

If the difference in the radius between the start and end points of the arc exceeds the value specified in parameter No. 3410, alarm PS0020 is generated.
If the end point is not on the arc, the tool moves in a straight line along one of the axes after reaching the end point.

Example

M



The above tool path can be programmed as follows ;

(1) In absolute programming

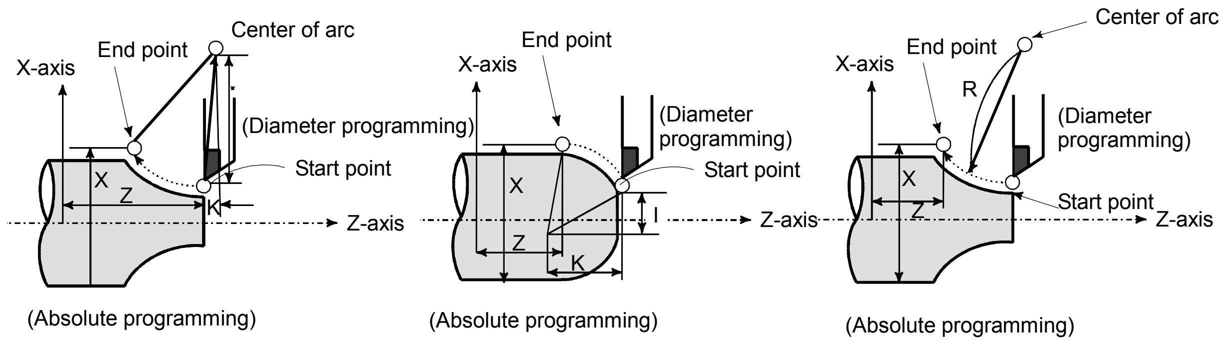
G92X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0R60.0 F300.;
G02 X120.0 Y60.0R50.0 ;
or
G92X200.0 Y40.0Z0 ;
G90 G03 X140.0 Y100.0I-60.0 F300.;
G02 X120.0 Y60.0I-50.0 ;

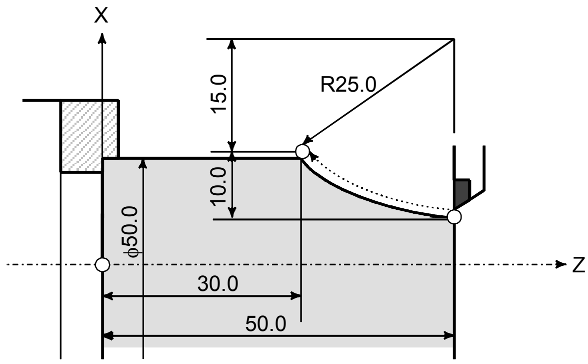
(2) In incremental programming

G91 G03 X-60.0 Y60.0 R60.0 F300.;
G02 X-20.0 Y-40.0 R50.0 ;
or
G91 G03 X-60.0 Y60.0 I-60.0 F300. ;
G02 X-20.0 Y-40.0 I-50.0 ;

T

- Command of circular interpolation X, Z





(Diameter programming)
G02X50.0Z30.0I25.0F0.3; or
G02U20.0W-20.0I25.0F0.3; or
G02X50.0Z30.0R25.0F0.3 or
G02U20.0W-20.0R25.F0.3;

4.5 HELICAL INTERPOLATION (G02, G03)

Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

Format

Arc of XpYp plane

$$\text{G17}\begin{Bmatrix}\text{G02}\\\text{G03}\end{Bmatrix}\text{Xp\_Yp\_}\begin{Bmatrix}\text{I\_J\_}\\\text{R\_}\end{Bmatrix}\alpha\_(\beta\_)\text{F\_;}$$

Arc of ZpXp plane

$$\text{G18}\begin{Bmatrix}\text{G02}\\\text{G03}\end{Bmatrix}\text{Zp\_Xp\_}\begin{Bmatrix}\text{K\_I\_}\\\text{R\_}\end{Bmatrix}\alpha\_(\beta\_)\text{F\_;}$$

Arc of YpZp plane

$$\text{G19}\begin{Bmatrix}\text{G02}\\\text{G03}\end{Bmatrix}\text{Yp\_Zp\_}\begin{Bmatrix}\text{J\_K\_}\\\text{R\_}\end{Bmatrix}\alpha\_(\beta\_)\text{F\_;}$$

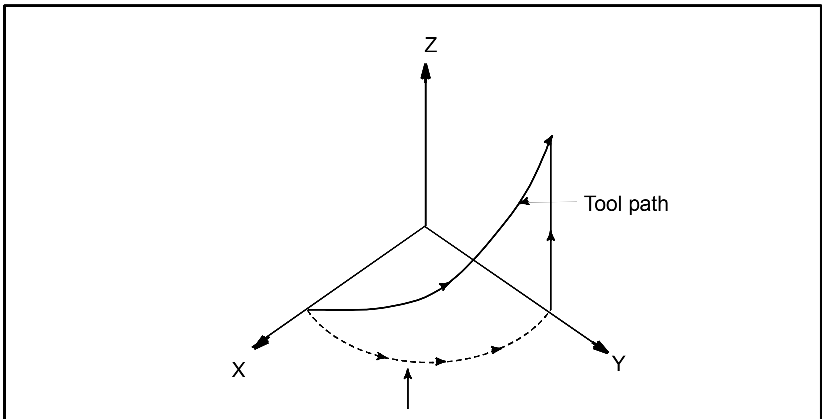
α, β: Any one axis where circular interpolation is not applied.
Up to two other axes can be specified.

Explanation

A tangential velocity of an arc in a specified plane or a tangential velocity about the linear axis can be specified as the feedrate, depending on the setting of bit 5 (HTG) of parameter No.1403.
An F command specifies a feedrate along a circular arc, when HTG is specified to 0. Therefore, the feedrate of the linear axis is as follows:

$$F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Determine the feedrate so the linear axis feedrate does not exceed any of the various limit values.

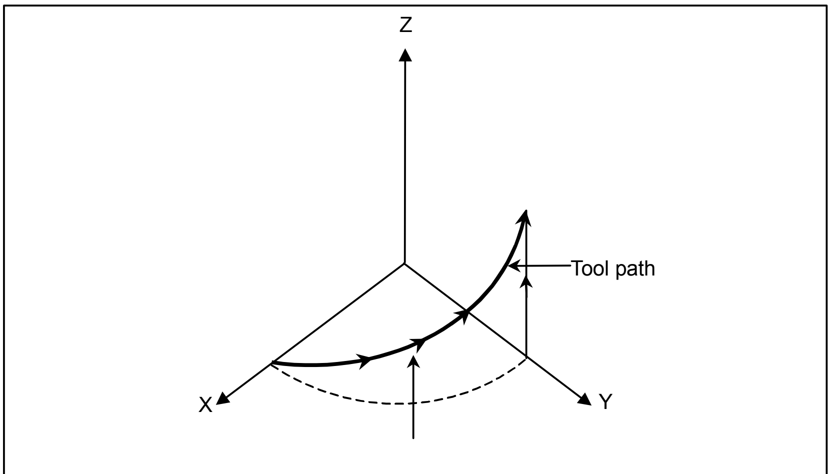


If HTG is set to 1, specify a feedrate along the tool path about the linear axis. Therefore, the tangential velocity of the arc is expressed as follows:

$$F \times \frac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$$

The velocity along the linear axis is expressed as follows:

$$F \times \frac{\text{Length of linear axis}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$$



Limitation

- Cutter compensation or tool nose radius compensation is applied only for a circular arc.
- Tool offset and tool length compensation cannot be used in a block in which a helical interpolation is commanded.

4.6 HELICAL INTERPOLATION B (G02, G03)

The helical interpolation B function differs from the helical interpolation function just in that circular interpolation and a movement on four axes outside the specified plane can be simultaneously performed.
For the restrictions and parameters, see the description of the helical interpolation function.

Format

Arc in the XpYp plane

$$\mathbf{G17}\begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Xp\_\_Yp\_\_}\ \begin{Bmatrix}\mathbf{I\_\_J\_\_}\\ \mathbf{R\_\_}\end{Bmatrix}\ \alpha\_\_\beta\_\_\gamma\_\_\delta\_\_\mathbf{F\_\_};$$

Arc in the ZpXp plane

$$\mathbf{G18}\begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Zp\_\_Xp\_\_}\ \begin{Bmatrix}\mathbf{K\_\_I\_\_}\\ \mathbf{R\_\_}\end{Bmatrix}\ \alpha\_\_\beta\_\_\gamma\_\_\delta\_\_\mathbf{F\_\_};$$

Arc in the YpZp plane

$$\mathbf{G19}\begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Yp\_\_Zp\_\_}\ \begin{Bmatrix}\mathbf{J\_\_K\_\_}\\ \mathbf{R\_\_}\end{Bmatrix}\ \alpha\_\_\beta\_\_\gamma\_\_\delta\_\_\mathbf{F\_\_};$$

α, β, γ, δ: Any axis to which circular interpolation is not applied.
Up to four axes can be specified.

4.7 SPIRAL INTERPOLATION, CONICAL INTERPOLATION (G02, G03)

Spiral interpolation is enabled by specifying the circular interpolation command together with a desired number of revolutions or a desired increment (decrement) for the radius per revolution.
Conical interpolation is enabled by specifying the spiral interpolation command together with an additional axis of movement, as well as a desired increment (decrement) for the position along the additional axes per spiral revolution.

Format

- Spiral interpolation

XpYp plane

$$\mathbf{G17}\ \left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{X\_Y\_I\_J\_Q\_L\_F\_;}$$

ZpXp plane

$$\mathbf{G18}\ \left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{Z\_X\_K\_I\_Q\_L\_F\_;}$$

YpZp plane

$$\mathbf{G19}\ \left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{Y\_Z\_J\_K\_Q\_L\_F\_;}$$

X, Y, Z : Coordinates of the end point
L : Number of revolutions (positive value without a decimal point) (*1)
Q : Radius increment or decrement per spiral revolution (*1, *2)
I, J, K : Signed distance from the start point to the center (same as the distance specified for circular interpolation)
F : Feedrate

(*1) Either the number of revolutions (L) or the radius increment or decrement (Q) can be omitted. When L is omitted, the number of revolutions is automatically calculated from the distance between the current position and the center, the position of the end point, and the radius increment or decrement. When Q is omitted, the radius increment or decrement is automatically calculated from the distance between the current position and the center, the position of the end point, and the number of revolutions. If both L and Q are specified but their values contradict, Q takes precedence. Generally, either L or Q should be specified. The L value must be a positive value without a decimal point. To specify four revolutions plus 90°, for example, round the number of revolutions up to five and specify L5.
(*2) The increment system for Q depends on the reference axis.

- Conical interpolation

XpYp plane

$$\textbf{G17}\left\{\begin{matrix}\textbf{G02}\\\textbf{G03}\end{matrix}\right\}\textbf{X\_Y\_I\_J\_Z\_Q\_L\_F\_;}$$

ZpXp plane

$$\textbf{G18}\left\{\begin{matrix}\textbf{G02}\\\textbf{G03}\end{matrix}\right\}\textbf{Z\_X\_K\_I\_Y\_Q\_L\_F\_;}$$

YpZp plane

$$\textbf{G19}\left\{\begin{matrix}\textbf{G02}\\\textbf{G03}\end{matrix}\right\}\textbf{Y\_Z\_J\_K\_X\_Q\_L\_F\_;}$$

X, Y, Z : Coordinates of the end point

L : Number of revolutions (positive value without a decimal point) (*1)

Q : Radius increment or decrement per spiral revolution (*1, *2)

I, J, K : Two of the three values represent a signed vector from the start point to the center. The remaining value is a height increment or decrement per spiral revolution in conical interpolation. (*1)

When the XpYp plane is selected:

The I and J values represent a signed vector from the start point to the center.

The K value represents a height increment or decrement per spiral revolution.

When the ZpXp plane is selected:

The K and I values represent a signed vector from the start point to the center.

The J value represents a height increment or decrement per spiral revolution.

When the YpZp plane is selected:

The J and K values represent a signed vector from the start point to the center.

The I value represents a height increment or decrement per spiral revolution.

F : Feedrate (The tangential velocity about the linear axis is specified.)

(*1) One of the height increment/decrement (I, J, K), radius increment/decrement (Q), and the number of revolutions (L) must be specified. The other two items can be omitted.

- Sample command for the XpYp plane

$$\textbf{G17}\left\{\begin{matrix}\textbf{G02}\\\textbf{G03}\end{matrix}\right\}\textbf{X\_Y\_I\_J\_Z\_}\left\{\begin{matrix}\textbf{K\_}\\\textbf{Q\_}\\\textbf{L\_}\end{matrix}\right\}\textbf{F\_;}$$

If both L and Q are specified, but their values contradict, Q takes precedence. If both L and a height increment or decrement are specified, but their values contradict, the height increment or decrement takes precedence. If both Q and a height increment or decrement are specified, but their values contradict, Q takes precedence. The L value must be a positive value without a decimal point. To specify four revolutions plus 90°, for example, round the number of revolutions up to five and specify L5.

(*2) The increment system for Q depends on the reference axis.

Explanation

- Function of spiral interpolation

Spiral interpolation in the XY plane is defined as follows:

$$(X - X_0)^2 + (Y - Y_0)^2 = (R + Q')^2$$

X_0: X coordinate of the center
Y_0: Y coordinate of the center
R : Radius at the beginning of spiral interpolation
Q' : Variation in radius

When the programmed command is assigned to this function, the following expression is obtained:

$$(X - X_s - I)^2 + (Y - Y_s - J)^2 = (R + (L' + \frac{\theta}{360})Q)^2$$

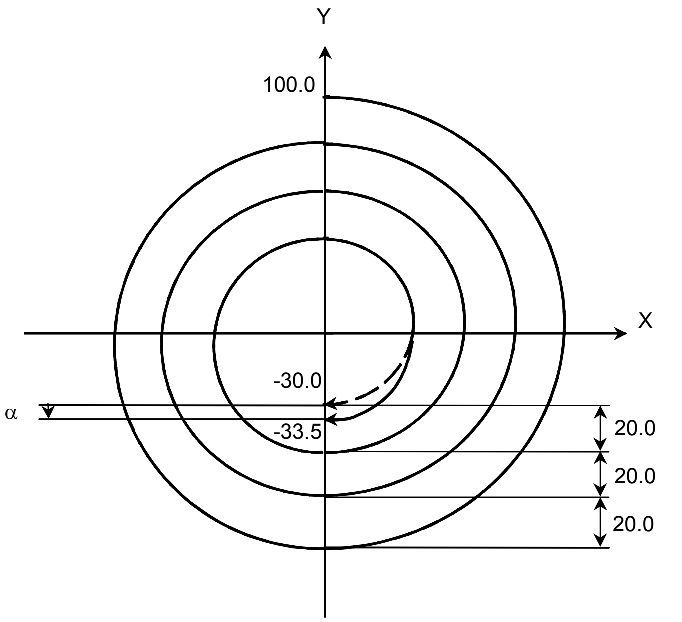
where
X_S: X coordinate of the start point
Y_S: Y coordinate of the start point
I : X coordinate of the vector from the start point to the center
J : Y coordinate of the vector from the start point to the center
R : Radius at the beginning of spiral interpolation
Q : Radius increment or decrement per spiral revolution
L' : (Current number of revolutions) - 1
θ : Angle between the start point and the current position (degrees)

- Controlled axes

For conical interpolation, two axes of a plane and two additional axes, that is, four axes in total, can be specified. A rotary axis can be specified as the additional axis.

- Difference between end points

If the difference between the programmed end point and the calculated end point of a spiral exceeds a value specified in parameter No. 3471 about any axis of a selected plane, an alarm PS5123 will be issued.
If the difference between the programmed height and calculated height of the end point a cone exceeds a value specified in parameter No. 3471, an alarm PS5123 will be issued. The figure below illustrates details.



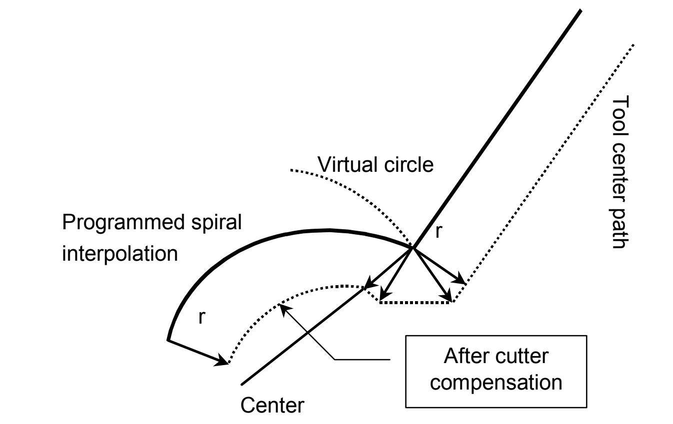
G90 G02 X0 Y-33.5 I0 J-100. { Q-20. / L4 } F300;

The coordinates of the programmed end point are (0, -33.5) while the coordinates of the calculated end point are (0, -30.0). A value greater than the difference (α: Tolerance) is specified in parameter No. 3471. If the end point is exceeded, an alarm PS5123 will be issued. The same is specified for the height of conical interpolation.

- Cutter compensation

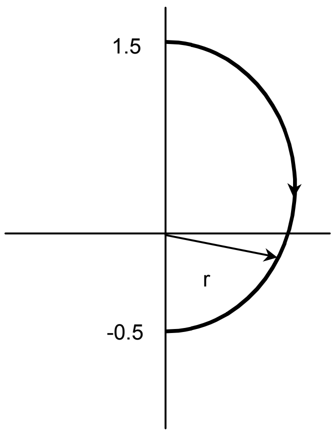
M

The spiral or conical interpolation command can be programmed in cutter compensation mode. This compensation is performed in the same way as described in "When it is exceptional" in "Tool Movement in Offset Mode" section. A virtual circle centered on the center of spiral interpolation is thought at the end of a block. The tool path is obtained by performing cutter compensation along the virtual circle and the blocks before and after the spiral interpolation. When the end point of the block is at the center of the spiral interpolation, no virtual circle can be drawn. If drawing is attempted, an alarm PS5124 is issued.



- Actual cutting feedrate

A constant speed is maintained in spiral interpolation or conical interpolation. The angular velocity near the center, however, may increase because of the small radius of the spiral. This can be avoided by maintaining the angular velocity after the radius of the spiral reaches a value specified in parameter No. 3472. Consequently, the actual cutting feedrate decreases.
An example is illustrated below.



Specified feedrate: F100.
Parameter 3472 (r) = 1.000
IS-B

While the radius of the spiral shown at the left is greater than the value specified in parameter No. 3472, the actual cutting feedrate is F100. As the radius decreases, the actual feedrate also decreases, the actual feedrate near the end point being about F65.

- Deceleration by acceleration

During spiral interpolation, the function of deceleration by acceleration is enabled. The feedrate may decrease as the tool approaches the center of the spiral.

- Dry run

When the dry run signal is inverted from 0 to 1 or from 1 to 0 during movement along an axis, the movement is accelerated or decelerated to the desired speed without first reducing the speed to zero.

Limitation

- Radius

T

In spiral or conical interpolation, R for specifying an arc radius and C or R for specifying optional-angle chamfering or corner rounding cannot be specified.

- Feed functions

The functions of feed per rotation, inverse time feed, F command with one digit, and automatic corner override cannot be used.

- Retrace

A program including spiral or conical interpolation cannot be retraced.

- Polar coordinate interpolation, scaling, and normal direction control

Spiral interpolation and conical interpolation cannot be specified in these modes.

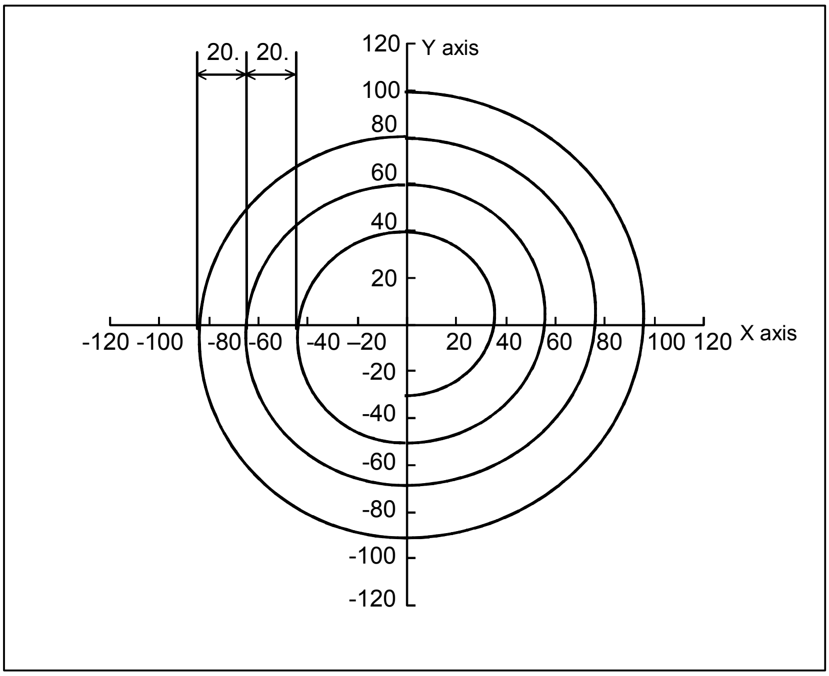
- Optional function

To use the conical interpolation function, the optional function for helical interpolation is also necessary.

Example

- Spiral interpolation

The path indicated below is programmed with absolute and incremental values as follows:



This sample path has the following values:

- Start point : (0, 100.0)
- End point (X, Y) : (0, -30.0)
- Distance to the center (I, J) : (0, -100.0)
- Radius increment or decrement (Q) : -20.0
- Number of revolutions (L) : 4.

(1) With absolute values, the path is programmed as follows:

$$\text{G90 G02 X0 Y-30.0 I0 J-100.0} \begin{Bmatrix} \text{Q-20.0} \\ \text{L4} \end{Bmatrix} \text{F300.;}$$

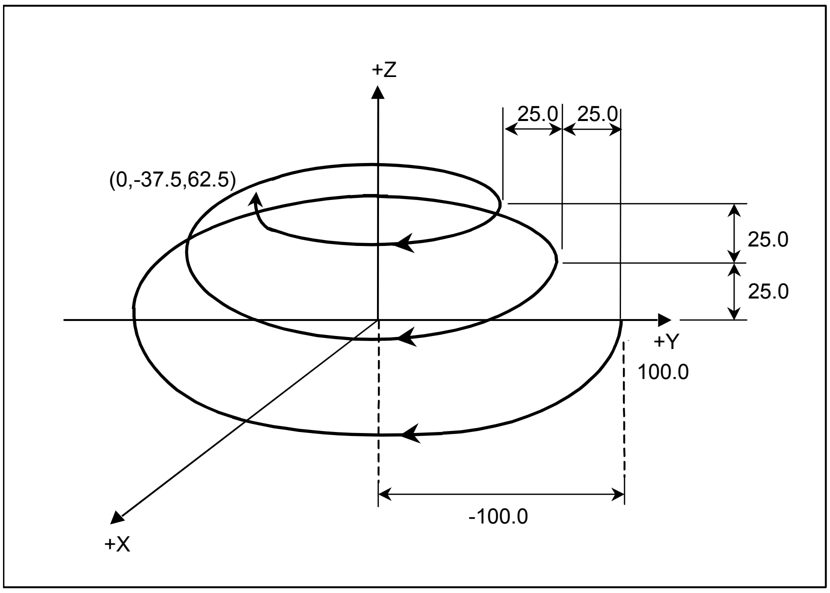
(2) With incremental values, the path is programmed as follows:

$$\text{G91 G02 X0 Y-130.0 I0 J-100.0} \begin{Bmatrix} \text{Q-20.0} \\ \text{L4} \end{Bmatrix} \text{F300.;}$$

(Either the Q or L setting can be omitted.)

- Conical interpolation

The sample path shown below is programmed with absolute and incremental values as follows:



This sample path has the following values:

- Start point : (0, 100.0, 0)
- End point (X, Y, Z) : (0, -37.5, 62.5)
- Distance to the center (I, J) : (0, -100.0)
- Radius increment or decrement (Q) : -25.0
- Height increment or decrement (K) : 25.0
- Number of revolutions (L) : 3

(1) With absolute values, the path is programmed as follows:

$$\text{G90 G02 X0 Y-37.5 Z62.5 I0 J-100.0} \quad \left\{\begin{matrix} \text{K25.0} \\ \text{Q-25.0} \\ \text{L3} \end{matrix}\right\} \quad \text{F300.;}$$

(2) With incremental values, the path is programmed as follows:

$$\text{G91 G02 X0 Y-137.5 Z62.5 I0 J-100.0} \quad \left\{\begin{matrix} \text{K25.0} \\ \text{Q-25.0} \\ \text{L3} \end{matrix}\right\} \quad \text{F300.;}$$

(Either the Q or L setting can be omitted.)

4.8 POLAR COORDINATE INTERPOLATION (G12.1, G13.1)

Overview

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This function is useful for grinding a cam shaft.

Format

G12.1;	Starts polar coordinate interpolation mode (enables polar coordinate interpolation)
⋮	Specify linear or circular interpolation using coordinates in a Cartesian coordinate system consisting of a linear axis and rotary axis (hypothetical axis).
G13.1	Polar coordinate interpolation mode is cancelled (for not performing polar coordinate interpolation)

Specify G12.1 and G13.1 in Separate Blocks.

Explanation

- Polar coordinate interpolation mode (G12.1)

The axes of polar coordinate interpolation (linear axis and rotary axis) should be specified in advance, with corresponding parameters. Specifying G12.1 places the system in the polar coordinate interpolation mode, and selects a plane (called the polar coordinate interpolation plane) formed by one linear axis and a hypothetical axis intersecting the linear axis at right angles. The linear axis is called the first axis of the plane, and the hypothetical axis is called the second axis of the plane. Polar coordinate interpolation is performed in this plane.

In the polar coordinate interpolation mode, both linear interpolation and circular interpolation can be specified by absolute or incremental programming.

Cutter compensation can also be performed. The polar coordinate interpolation is performed for a path obtained after cutter compensation.

The tangential velocity in the polar coordinate interpolation plane (Cartesian coordinate system) is specified as the feedrate, using F.

- Polar coordinate interpolation cancel mode (G13.1)

Specifying G13.1 cancels the polar coordinate interpolation mode.

- Polar coordinate interpolation plane

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 4.8 (a)). Polar coordinate interpolation is performed on this plane.

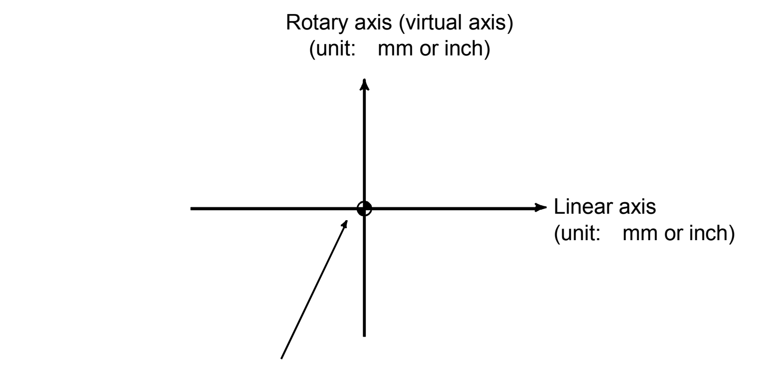


Fig. 4.8 (a) Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters (No. 5460 and 5461) beforehand.

> ⚠ **CAUTION**
> The plane used before G12.1 is specified (plane selected by G17, G18, or G19) is canceled. It is restored when G13.1 (canceling polar coordinate interpolation) is specified.
> When the system is reset, polar coordinate interpolation is canceled and the plane specified by G17, G18, or G19 is used.

- Distance moved and feedrate for polar coordinate interpolation

- The unit for coordinates on the hypothetical axis is the same as the unit for the linear axis (mm/inch).
  In the polar coordinate interpolation mode, program commands are specified with Cartesian coordinates on the polar coordinate interpolation plane. The axis address for the rotary axis is used as the axis address for the second axis (hypothetical axis) in the plane. Whether a diameter or radius is specified for the first axis in the plane is the same as for the rotary axis regardless of the specification for the first axis in the plane.
  The hypothetical axis is at coordinate 0 immediately after G12.1 is specified. Polar interpolation is started assuming the rotation angle of 0 for the position of the tool when G12.1 is specified.

Example)

When a value on the X-axis (linear axis) is input in millimeters

G12.1;

G01 X10. F1000.;..... A 10-mm movement is made on the Cartesian coordinate system.

C20.;......................... A 20-mm movement is made on the Cartesian coordinate system.

G13.1;

When a value on the X-axis (linear axis) is input in inches

G12.1;

G01 X10. F1000.; ... A 10-inch movement is made on the Cartesian coordinate system.

C20.;......................... A 20-inch movement is made on the Cartesian coordinate system.

G13.1;

- The unit for the feedrate is mm/min or inch/min.
  Specify the feedrate as a speed (relative speed between the workpiece and tool) tangential to the polar coordinate interpolation plane (Cartesian coordinate system) using F.

- G codes which can be specified in the polar coordinate interpolation mode

G01 ....Linear interpolation
G02, G03...............Circular interpolation
G02.2, G03.2.........Involute interpolation
G04 ....Dwell, Exact stop
G40, G41, G42......Cutter compensation
(Polar coordinate interpolation is applied to the path after cutter compensation.)
G65, G66, G67......Custom macro command
G90, G91...............Absolute command, incremental command
G94, G95...............Feed per minute, feed per revolution

- Circular interpolation in the polar coordinate plane

The addresses for specifying the radius of an arc for circular interpolation (G02 or G03) in the polar coordinate interpolation plane depend on the first axis in the plane (linear axis).

- I and J in the Xp-Yp plane when the linear axis is the X-axis or an axis parallel to the X-axis.
- J and K in the Yp-Zp plane when the linear axis is the Y-axis or an axis parallel to the Y-axis.
- K and I in the Zp-Xp plane when the linear axis is the Z-axis or an axis parallel to the Z-axis.

The radius of an arc can be specified also with an R command.

NOTE
In a lathe system, the parallel axes U, V, and W can be used in the G code system B or C.

- Movement along axes not in the polar coordinate interpolation plane in the polar coordinate interpolation mode

The tool moves along such axes normally, independent of polar coordinate interpolation.

- Current position display in the polar coordinate interpolation mode

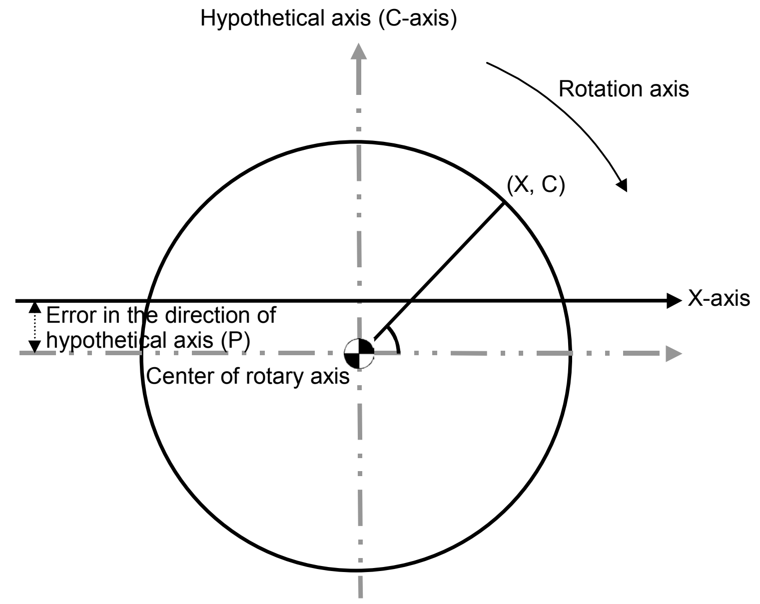
Actual coordinates are displayed. However, the remaining distance to move in a block is displayed based on the coordinates in the polar coordinate interpolation plane (Cartesian coordinates).

- Coordinate system for the polar coordinate interpolation

Basically, before G12.1 is specified, a local coordinate system (or workpiece coordinate system) where the center of the rotary axis is the origin of the coordinate system must be set.

- Compensation in the direction of the hypothetical axis in polar coordinate interpolation

If the first axis of the plane has an error from the center of the rotary axis in the hypothetical axis direction, in other words, if the rotary axis center is not on the X-axis, the hypothetical axis direction compensation function in the polar coordinate interpolation mode is used. With the function, the error is considered in polar coordinate interpolation. The amount of error is specified in parameter No. 5464.

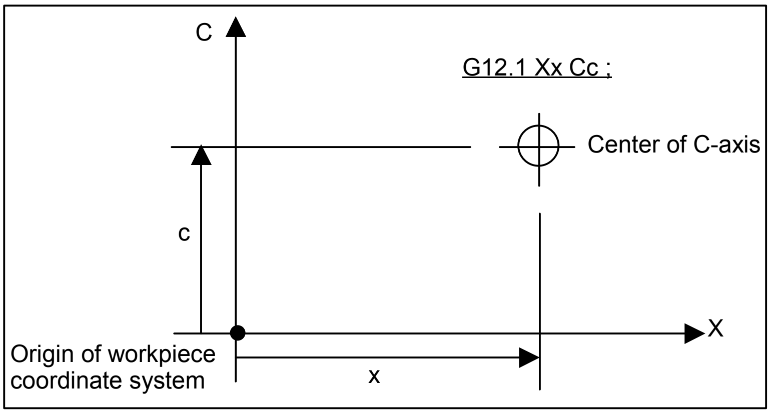


(X, C)	Point in the X-C plane (The center of the rotary axis is considered to be the origin of the X-C plane.)
X	X coordinate in the X-C plane
C	Hypothetical axis coordinate in the X-C plane
P	Error in the direction of the hypothetical axis (specified in parameter No. 5464)

- Shifting the coordinate system in polar coordinate interpolation

In the polar coordinate interpolation mode, the workpiece coordinate system can be shifted. The current position display function shows the position viewed from the workpiece coordinate system before the shift. The function to shift the coordinate system is enabled when bit 2 of parameter No. 5450 is specified accordingly.
The shift can be specified in the polar coordinate interpolation mode, by specifying the position of the center of the rotary axis C (A, B) in the X-C (Y-A, Z-B) interpolation plane with reference to the origin of the workpiece coordinate system, in the following format.

G12.1 X_ C_ ;	(Polar coordinate interpolation for the X-axis and C-axis)
G12.1 Y_ A_ ;	(Polar coordinate interpolation for the Y-axis and A-axis)
G12.1 Z_ B_ ;	(Polar coordinate interpolation for the Z-axis and B-axis)



Limitation

- Changing the coordinate system during polar coordinate interpolation

In the G12.1 mode, the coordinate system must not be changed (G92, G52, G53, relative coordinate reset, G54 through G59, etc.).

- Cutter or tool nose radius compensation

The polar coordinate interpolation mode (G12.1 or G13.1) cannot be started or terminated in the cutter or tool nose radius compensation mode (G41 or G42). G12.1 or G13.1 must be specified in the cutter or tool nose radius compensation canceled mode (G40).

- Tool length offset command

Tool length offset must be specified in the polar coordinate interpolation cancel mode before G12.1 is specified. It cannot be specified in the polar coordinate interpolation mode. Furthermore, no offset values can be changed in the polar coordinate interpolation mode.

- Tool offset command

A tool offset must be specified before the G12.1 mode is set. No offset can be changed in the G12.1 mode.

- Program restart

For a block in the G12.1 mode, the program and the block cannot be restarted.

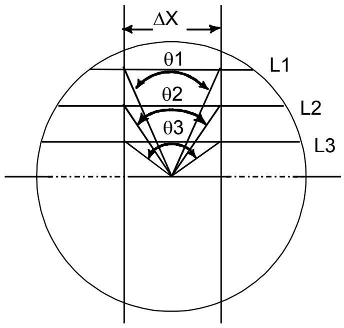
- Cutting feedrate for the rotary axis

Polar coordinate interpolation converts the tool movement for a figure programmed in a Cartesian coordinate system to the tool movement in the rotary axis (C-axis) and the linear axis (X-axis). When the tool comes close to the center of the workpiece, the C-axis velocity component increases. If the maximum cutting feedrate for the C-axis (parameter No. 1430) is exceeded, the automatic feedrate override function and automatic speed clamp function are enabled.

If the maximum cutting feedrate for the X-axis is exceeded, the automatic feedrate override function and automatic speed clamp function are enabled.

⚠ WARNING

1 Consider lines L1, L2, and L3. ΔX is the distance the tool moves per time unit at the feedrate specified with address F in the Cartesian coordinate system. As the tool moves from L1 to L2 to L3, the angle at which the tool moves per time unit corresponding to ΔX in the Cartesian coordinate system increases from θ1 to θ2 to θ3. In other words, the C-axis component of the feedrate becomes larger as the tool moves closer to the center of the workpiece. The C component of the feedrate may exceed the maximum cutting feedrate for the C-axis because the tool movement in the Cartesian coordinate system has been converted to the tool movement for the C-axis and the X-axis.



L: Distance (in mm) between the tool center and workpiece center when the tool center is the nearest to the workpiece center

R: Maximum cutting feedrate (deg/min) of the C axis

Then, a speed specifiable with address F in polar coordinate interpolation can be given by the formula below. If the maximum cutting feedrate for the C-axis is exceeded, the automatic speed control function for polar coordinate interpolation automatically controls the feedrate.

$$F < L \times R \times \frac{\pi}{180} \text{ (mm/min)}$$

2 The following function cannot be used for the rotary axis of polar coordinate interpolation.

- Index table indexing function

- Automatic speed control for polar coordinate interpolation

If the velocity component of the rotary axis exceeds the maximum cutting feedrate in the polar coordinate interpolation mode, the speed is automatically controlled.

- Automatic override

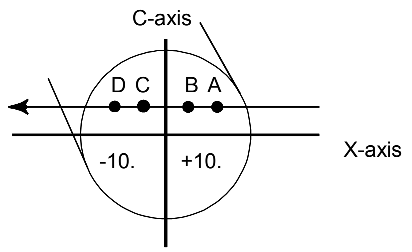
If the velocity component of the rotary axis exceeds the permissible velocity (maximum cutting feedrate multiplied by the permission factor specified in parameter No. 5463), the feedrate is automatically overridden as indicated below.
Override = (Permissible velocity) ÷ (Velocity component of rotary axis) × 100(%)

- Automatic speed clamp

If the velocity component of the rotary axis after automatic override still exceeds the maximum cutting feedrate, the speed of the rotary axis is automatically clamped. As a result, the velocity component of the rotary axis will not exceed the maximum cutting feedrate.
The automatic speed clamp function works only when the center of the tool is very close to the center of the rotary axis.

[Example]

```
G90 G00 X10.0 C0. ;
G12.1 ;
G01 C0.1 F1000 ;
X-10.0 :
G13.1 ;
```



Automatic speed control for polar coordinate interpolation

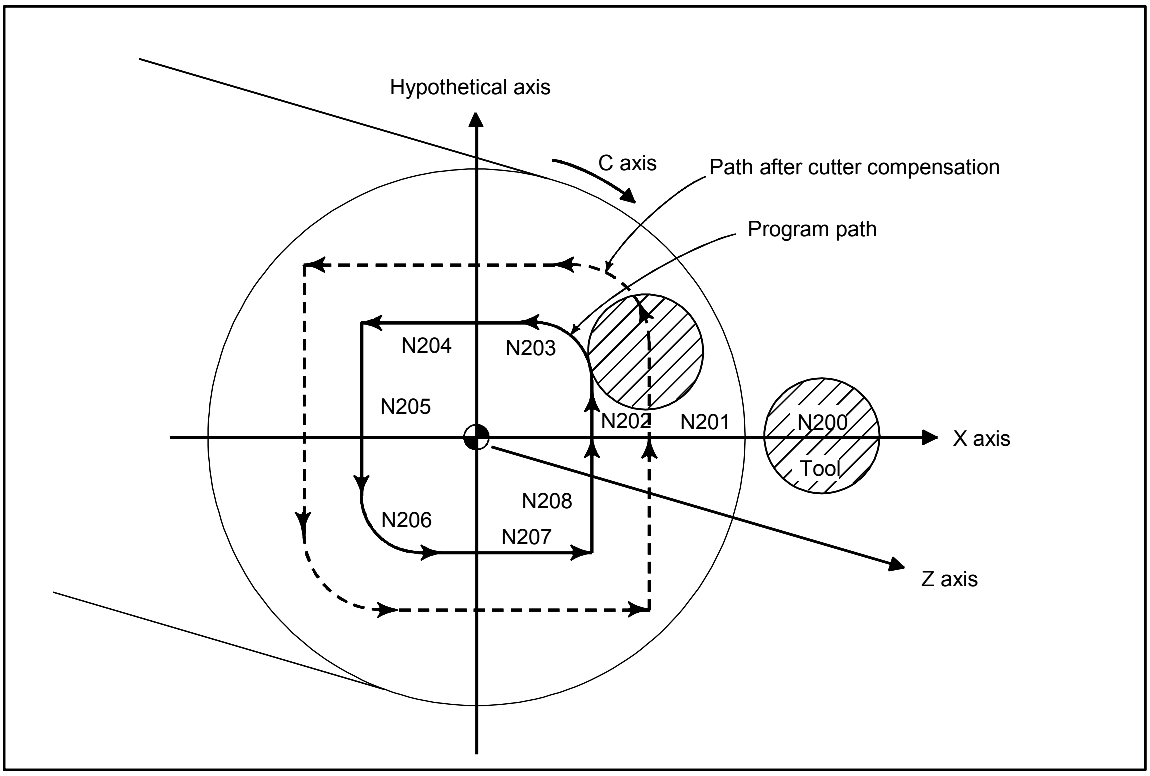
Suppose that the maximum cutting feedrate of the rotary axis is 360 (3600 deg/min) and that the permission factor of automatic override for polar coordinate interpolation (parameter No. 5463) is 0 (90%). If the program indicated above is executed, the automatic override function starts working when the X coordinate becomes 2.273 (point A). The automatic speed clamp function starts working when the X coordinate becomes 0.524 (point B).
The minimum value of automatic override for this example is 3%. The automatic speed clamp function continues working until the X coordinate becomes -0.524 (point C). Then, the automatic override function works until the X coordinate becomes -2.273 (point D).
(The coordinates indicated above are the values in the Cartesian coordinate system.)

NOTE

1. While the automatic speed clamp function is working, the machine lock or interlock function may not be enabled immediately.
2. If a feed hold stop is made while the automatic speed clamp function is working, the automatic operation halt signal is output. However, the operation may not stop immediately.
3. The clamped speed may exceed the clamp value by a few percent.

Example

Sample program for polar coordinate interpolation in a Cartesian coordinate system consisting of the X-axis (a linear axis) and a hypothetical axis




```
O001;
   .
No10 T0101
   .
N0100 G90 G00 X60.0 C0 Z__;          Positioning to start point
N0200 G12.1;                         Start of polar coordinate interpolation
N0201 G42 G01 X20.0F__;
N0202 C10.0;
N0203 G03 X10.0 C20.0 R10.0;
N0204 G01 X-20.0;                    Geometry program
N0205 C-10.0;                        (program based on cartesian
N0206 G03 X-10.0 C-20.0 I10.0 J0;    coordinates on X
N0207 G01 X20.0;                     axis-hypothetical axis plane)
N0208 C0;
N0209 G40 X60.0;
N0210 G13.1;                         Cancellation of polar coordinate
N0300 Z__:                           interpolation
N0400 X__C__;
   .
N0900M30;
```

4.9 CYLINDRICAL INTERPOLATION (G07.1)

In cylindrical interpolation, the amount of movement of a rotary axis specified by angle is converted to the amount of movement on the circumference to allow linear interpolation and circular interpolation with another axis.
Since programming is enabled with the cylinder side face expanded, programs such as a program for grooving cylindrical cams can be created very easily.

Format

G07.1 IP r; Starts the cylindrical interpolation mode (enables cylindrical interpolation).
:
:
:
G07.1 IP 0; The cylindrical interpolation mode is cancelled.
IP : An address for the rotary axis
r : The radius of the workpiece
Specify G07.1 IPr ; and G07.1 IP0; in separate blocks.
G107 can be used instead of G07.1.

Explanation

- Plane selection (G17, G18, G19)

To specify a G code for plane selection, set the rotary axis in parameter No. 1022 as a linear axis that is one of the basic three axes of the basic coordinate system or an axis parallel to one of the basic axes. For example, when rotary axis C-axis is assumed to be parallel to the X-axis, specifying G17, axis address C, and Y at the same time can select a plane formed by the C-axis and Y-axis (the Xp-Yp plane).

T

NOTE
The U-, V-, and W-axes can be used with G-codes B and C.

- Feedrate

A feedrate specified in the cylindrical interpolation mode is the feedrate on the circumference.

- Circular interpolation (G02,G03)

Circular interpolation can be performed between the rotary axis set for cylindrical interpolation and another linear axis. Radius R is used in commands in the same way as described.
The unit for a radius is not degrees but millimeters (for metric input) or inches (for inch input).

<Example Circular interpolation between the Z axis and C axis>
For the C axis of parameter (No.1022), 5 (axis parallel with the X axis) is to be set. In this case, the command for circular interpolation is
G18 Z_C_;
G02 (G03) Z_C_R_;
For the C axis of parameter (No.1022), 6 (axis parallel with the Y axis) may be specified instead. In this case, however, the command for circular interpolation is
G19 C_Z_;
G02 (G03) Z_C_R_;

- Cutter or tool nose radius compensation

To perform cutter or tool nose radius compensation in the cylindrical interpolation mode, cancel any ongoing cutter or tool nose radius compensation mode before entering the cylindrical interpolation mode. Then, start and terminate cutter or tool nose radius compensation within the cylindrical interpolation mode.

- Cylindrical interpolation accuracy

In the cylindrical interpolation mode, the amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis on the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to an angle. For this conversion, the amount of travel is rounded to a least input increment.
So when the radius of a cylinder is small, the actual amount of travel can differ from a specified amount of travel. Note, however, that such an error is not accumulative.
If manual operation is performed in the cylindrical interpolation mode with manual absolute on, an error can occur for the reason described above.

$$\text{The actual amount of travel} = \left[\frac{\text{MOTION REV}}{2\times 2\pi \text{R}}\times\left[\text{Specified value}\times\frac{2\times 2\pi \text{R}}{\text{MOTION REV}}\right]\right]$$

MOTION REV: The amount of travel per rotation of the rotary axis (360°)
R: Workpiece radius
[ ]: Rounded to the least input increment

Limitation

- Arc radius specification in the circular interpolation

In the cylindrical interpolation mode, an arc radius cannot be specified with word address I, J, or K.

- Positioning

In the cylindrical interpolation mode, positioning operations (including those that produce rapid traverse cycles such as G28, G53, G73, G74, G76, G80 to G89) cannot be specified. Before positioning can be specified, the cylindrical interpolation mode must be cancelled. Cylindrical interpolation (G07.1) cannot be performed in the positioning mode (G00).

- Cylindrical interpolation mode setting

In the cylindrical interpolation mode, the cylindrical interpolation mode cannot be reset. The cylindrical interpolation mode must be cancelled before the cylindrical interpolation mode can be reset.

- Rotary axis

Only one rotary axis can be set for cylindrical interpolation. Therefore, it is impossible to specify more than one rotary axis in the G07.1 command.

- Rotary axis roll-over

If a rotary axis using the roll-over function is specified at the start of the cylindrical interpolation mode, the roll-over function is automatically disabled in the cylindrical interpolation mode. After the cylindrical interpolation mode is canceled, the roll-over function is enabled automatically.

- Rotary axis control function

If a rotary axis using the multiple rotary axis control function is specified at the start of the cylindrical interpolation mode, the multiple rotary axis control function is automatically disabled in the cylindrical interpolation mode. After the cylindrical interpolation mode is canceled, the multiple rotary axis control function is enabled automatically.

- Cutter or tool nose radius compensation

If the cylindrical interpolation mode is specified when cutter or tool nose radius compensation is already being applied, correct compensation is not performed. Specify compensation in the cylindrical interpolation mode.

- Canned cycle for drilling

Canned cycles (G73, G74, and G81 to G89 for M series / G80 to G89 for T series) for drilling, cannot be specified during cylindrical interpolation mode.

M

- Coordinate system setting

In the cylindrical interpolation mode, a workpiece coordinate system (G92, G54 to G59) or local coordinate system (G52) cannot be specified.

- Tool offset

A tool offset must be specified before the cylindrical interpolation mode is set. No offset can be changed in the cylindrical interpolation mode.

- Index table indexing function

Cylindrical interpolation cannot be specified when the index table indexing function is being used.

- Parallel axis

The rotary axis specified for cylindrical interpolation must not be a parallel axis.

T

- Coordinate system setting

In the cylindrical interpolation mode, a workpiece coordinate system G50 cannot be specified.

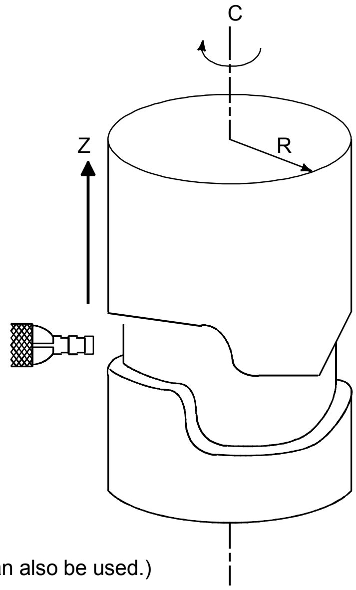
- Mirror image for double turret

Mirror image for double turret, G68 and G69, cannot be specified during cylindrical interpolation mode.

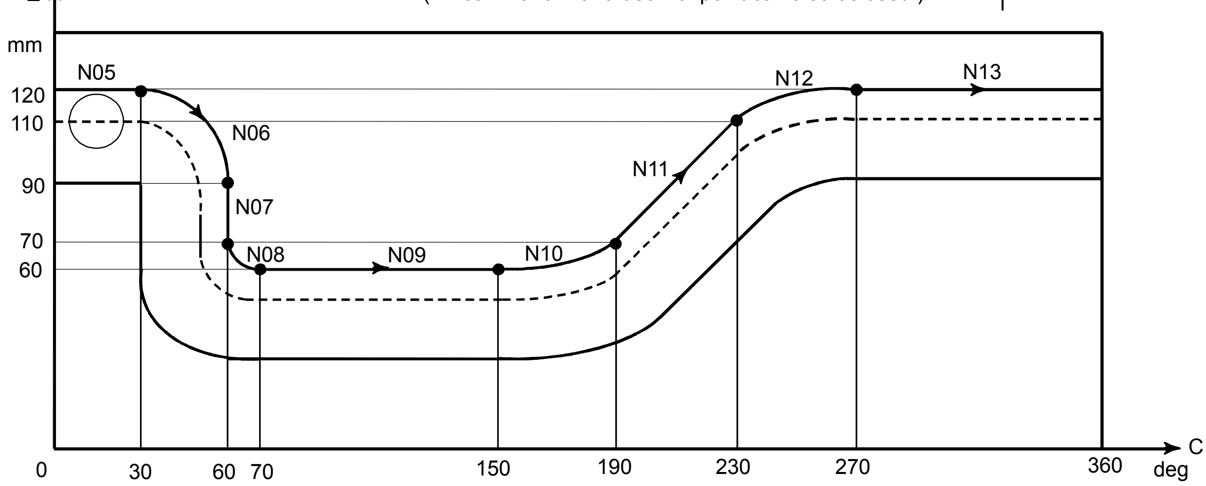
Example

Example of a Cylindrical Interpolation

```
O0001 (CYLINDRICAL INTERPOLATION );
N01 G00 G90 Z100.0 C0 ;
N02 G01 G91 G18 Z0 C0 ;
N03 G07.1 C57299 ;*
N04 G90 G01 G42 Z120.0 D01 F250 ;
N05 C30.0 ;
N06 G03 Z90.0 C60.0 R30.0 ;
N07 G01 Z70.0 ;
N08 G02 Z60.0 C70.0 R10.0 ;
N09 G01 C150.0 ;
N10 G02 Z70.0 C190.0 R75.0 ;
N11 G01 Z110.0 C230.0 ;
N12 G03 Z120.0 C270.0 R75.0 ;
N13 G01 C360.0 ;
N14 G40 Z100.0 ;
N15 G07.1 C0 ;
N16 M30 ;
```



(* A command with a decimal point can also be used.)



4.10 CUTTING POINT INTERPOLATION FOR CYLINDRICAL INTERPOLATION (G07.1)

The conventional cylindrical interpolation function controls the tool center so that the tool axis always moves along a specified path on the cylindrical surface, towards the rotary axis (cylindrical axis) of the workpiece. The cutting point interpolation for cylindrical interpolation function controls the tool so that the tangent of the tool and the cutting surface of a contour always passes the rotation center of the workpiece. This means that the cutting surface of the contour is always perpendicular to the cylinder. With this function, the figure on the cutting surface can always be kept constant regardless of the cutter compensation value of the tool used.

Format

This command is specified in the same way as for the conventional cylindrical interpolation function.

G07.1 IPr ; Circular interpolation mode on start (enabling cylindrical interpolation)

:

G07.1 IP0 ; Circular interpolation mode cancel

IP : One rotary axis address

r : Cylinder radius of rotary axis

Specify each of G07.1 IPr; and G07.1 IP0; singly in a block.

G107 can also be specified.

Explanation

- Comparison with conventional circular interpolation

As shown in Fig. 4.10 (a), the tool is controlled in the offset axis (Y-axis) direction that is perpendicular to the tool center and workpiece rotation center.

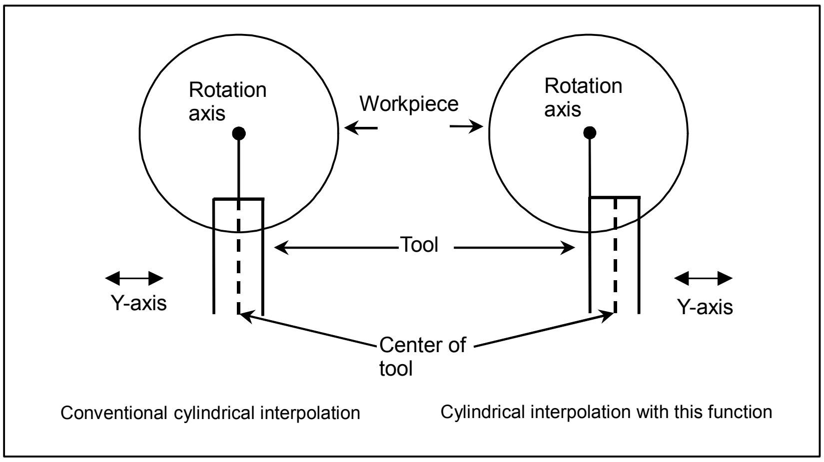


Fig. 4.10 (a) Comparison with conventional circular interpolation

- Cutting point compensation

(1) Cutting point compensation between blocks
As shown in Fig. 4.10(b), cutting point compensation is achieved by moving between blocks N1 and N2.
(a) Let C1 and C2 be the heads of the vectors normal to N1 and N2 from S1, which is the intersection of the tool center paths of blocks N1 and N2
(b) After the tool moves to S1 according to the command of N1, the tool moves through V on the C-axis as a result of cutting point compensation, then through $-V \times \frac{\pi}{180} \times r$ along the Y-axis.

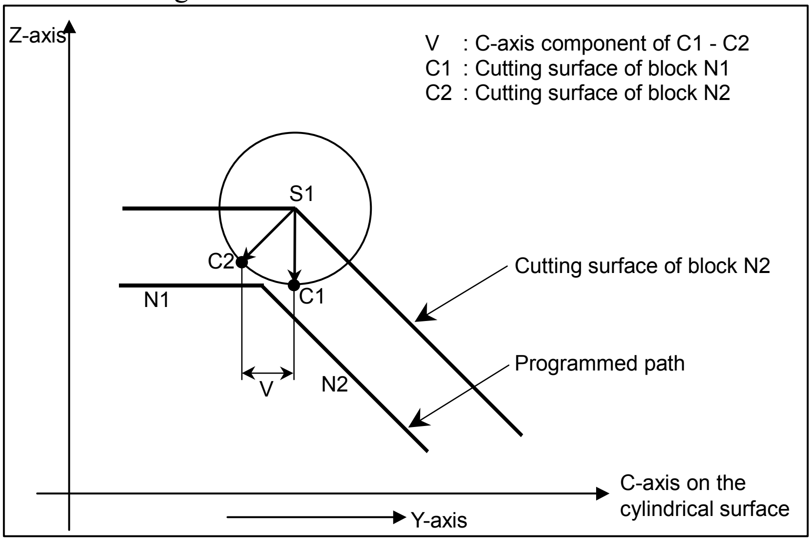


Fig.4.10 (b) Cutting point compensation between blocks

(2) Cutting point compensation in a circular command block
As shown in Fig. 4.10(c), the movement required for cutting point compensation is made simultaneously with circular interpolation in block N1.
(a) Let C0 be the head of the vector normal to N1 from S0, which is the tool center position at the start point of circular block N1. Let C1 be the head of the similar vector at the end point.
(b) As the tool moves from S0 to S1, a superimposed movement is made by the C-axis component of (C1 - C2) (V in the figure) on the C-axis, and a superimposed movement is made by $-V \times \frac{\pi}{180} \times r$ along the Y-axis. along the Y-axis.
That is, the following expressions are valid. As movement is made through L as shown in Fig. 4.10(c), the superimposed movements are made on the C-axis and Y-axis as follows:

$$\Delta C = \Delta V$$

$$\Delta Y = -\frac{\pi}{180}(\Delta V)r$$

ΔV : Cutting point compensation value (ΔV2 - ΔV1) for movement of ΔL

ΔV1 : C-axis component of the vector normal to N1 from the tool center of the start point of ΔL

ΔV2 : C-axis component of the vector normal to N1 from the tool center of the end point of ΔL

R : Arc radius

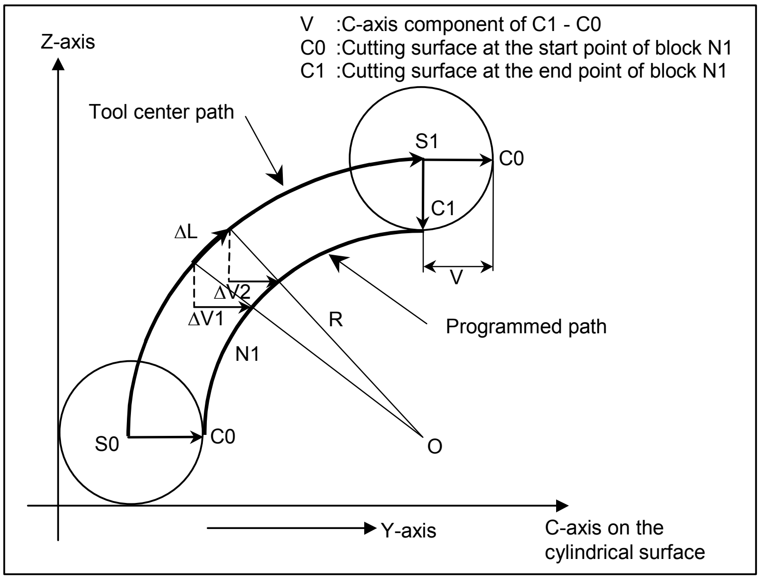


Fig.4.10 (c) Cutting point compensation in a circular command block

(3) When cutting point compensation is not applied between blocks
When, as shown in Fig. 4.10(d) and Fig. 4.10(e), the cutting point compensation value (V in the figures) is less than the value set in parameter No. 19534, one of the operations below is performed. (The operation that is performed depends on the setting of parameter CYS (No.19530#6).

(a) When parameter CYS (No.19530#6) is set to 1
Cutting point compensation is not applied between blocks N1 and N2, but is applied when block N2 is executed.

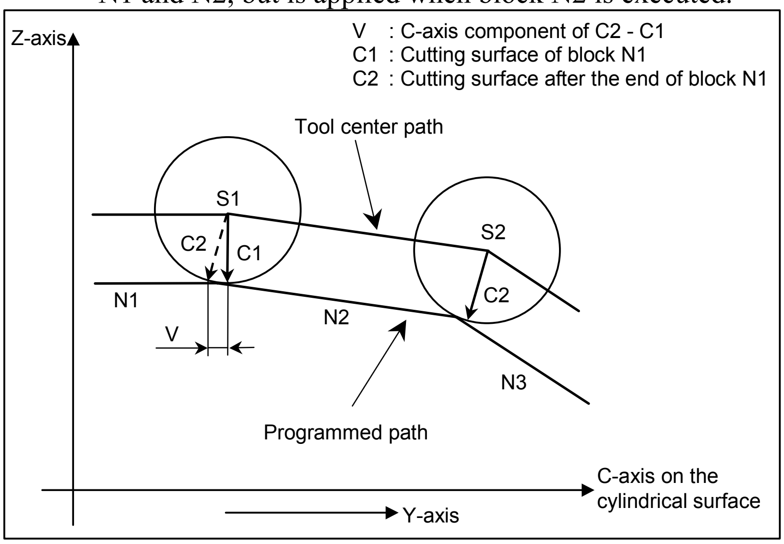


Fig.4.10 (d) When parameter CYS (No.19530#6) is set to 1

(b) When parameter CYS (No.19530#6) is set to 0
Cutting point compensation is not performed between blocks N1 and N2. Whether to apply cutting point compensation between block N2 and N3 is determined by taking the cutting point compensation value between blocks N2 and N3 (V in the figure) into consideration.

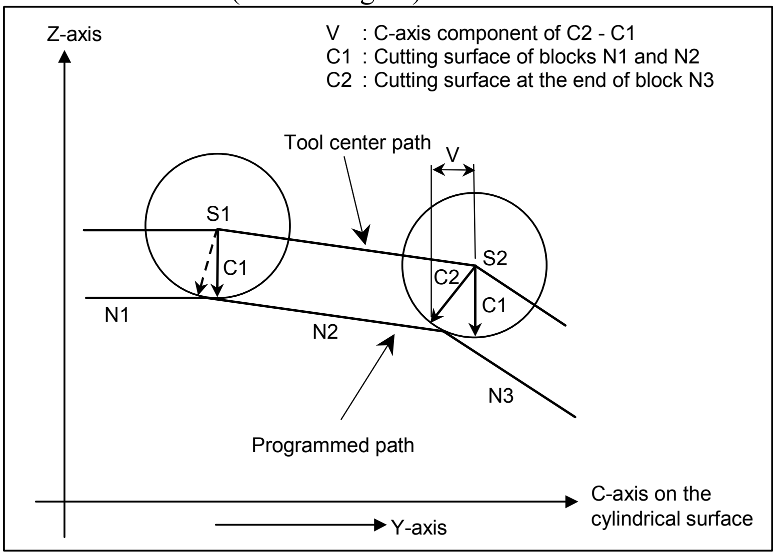


Fig.4.10 (e) When parameter CYS (No.19530#6) is 0

(b) When the amount of travel (L1) of block N2 is less than the value set in parameter No. 19535, as shown in Fig. 4.10(f), cutting point compensation is not applied between blocks N1 and N2. Instead, block N2 is executed with the cutting point compensation of the previous block. When the amount of travel (L2) of block N3 is greater than the value set in parameter No. 19535, cutting point compensation is applied between blocks N2 and N3.

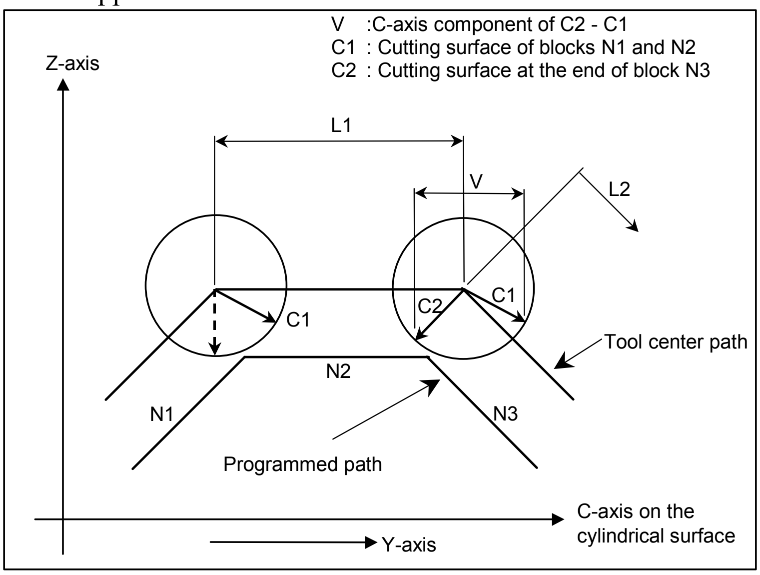


Fig.4.10 (f) When the amount of travel (L1) of block N2 is less than the parameter value

(d) When, as shown in Fig. 4.10(g), the diameter of an arc (R in the figure) is less than the value set in parameter No. 19535, cutting point compensation is not applied simultaneously with circular interpolation

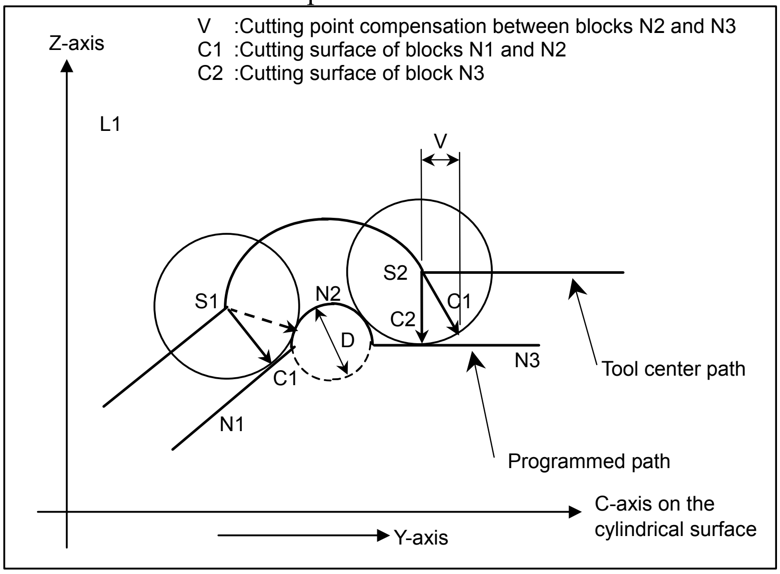


Fig.4.10 (g) When the diameter of an arc is less than the parameter value

- When cutting point compensation is used with normal direction control

When cutting point compensation is used with normal direction control, cutting point compensation between specified blocks is performed regardless of the method described in "**Cutting point compensation**" above, but is performed simultaneously with the movement of the normal direction control axis (C-axis).

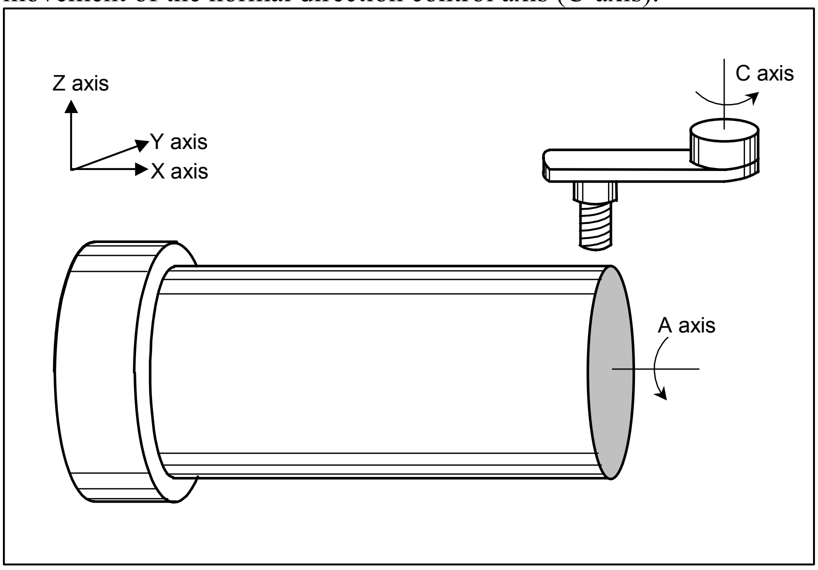


Fig. 4.10 (h) When used with normal direction control

(1) When the normal direction changes between blocks N1 and N2, cutting point compensation is also performed between blocks N1 and N2.
As shown in Fig. 4.10 (i), cutting point compensation described in (1) in "**Cutting point compensation**" is performed simultaneously with the movement made by normal direction control between blocks N1 and N2.

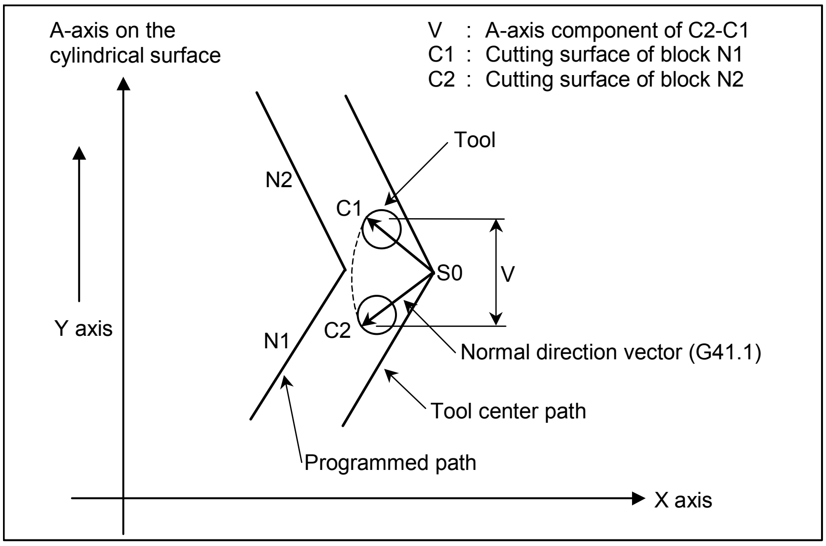


Fig. 4.10 (i) When the normal direction changes between blocks N1 and N2

(2) When gentle normal direction control changes the normal direction as a specified block is executed, cutting point compensation is performed simultaneously with the movement made by the block. When the normal direction control axis rotates by θ1 simultaneously with the movement made by blocks N1 to N2 as shown in Fig. 4.10 (j), cutting point compensation by the movement of vector V1 is also performed simultaneously with the movement in N2.

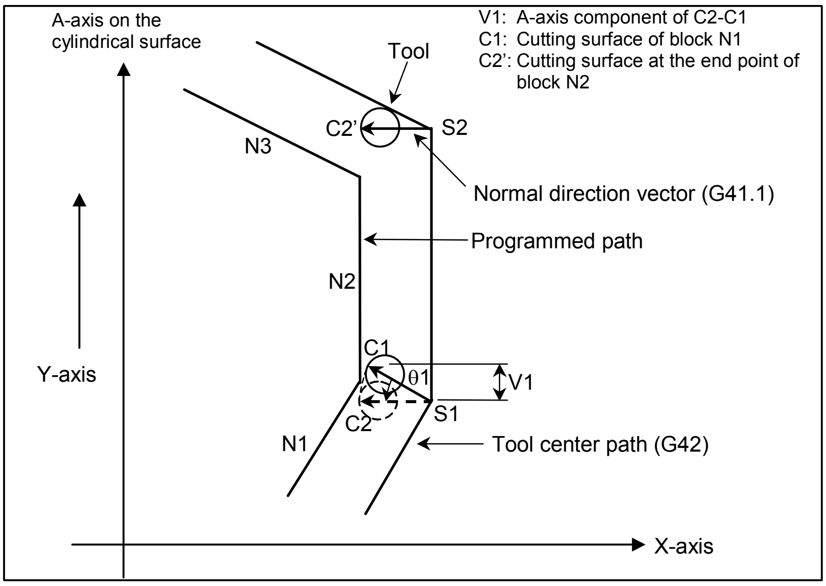


Fig. 4.10 (j) Gentle normal direction control

(3) When a specified block is executed while the normal direction control axis is held in the normal direction set at the end point of the previous block, cutting point compensation is not performed, and cutting point compensation applied in the previous block is maintained. As shown in Fig. 4.10 (k), when the movement amount in N2 (L1 in the figure) is smaller than the setting of parameter No. 5483, so the normal direction control axis does not rotate at S1 while the movement amount in N3 (L2 in the figure) is greater than the setting of parameter No. 5483, so the normal direction axis rotates at S2, cutting point compensation is not performed at S1 but is performed by the movement for vector V2 at S2.

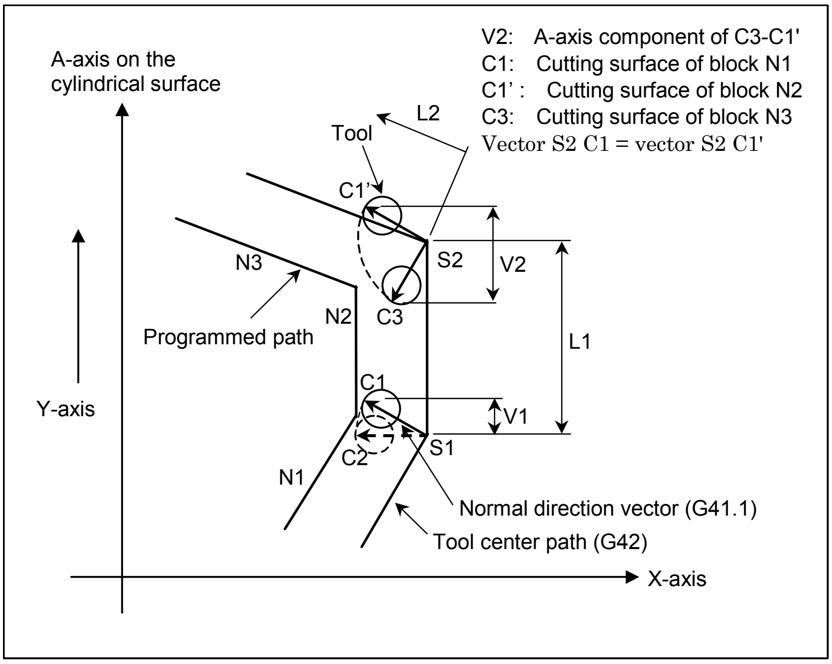


Fig. 4.10 (k) When the normal direction is the same as in the previous block

- Feedrate during cutting point compensation

(1) The tool moves at a specified feedrate while cutting point compensation is being applied between blocks.

(2) The actual speed indication and feedrate during circular interpolation are as described below.

Actual speed indication

The speed component of each axis after cutting point compensation at a point in time during circular interpolation is as follows:

$Fz' = Fz$: Speed component of linear axis

$Fc' = Fc + (Vce - Vcs)$: Speed component of rotary axis

$Fy' = -(Vce - Vcs)\dfrac{\pi r}{180}$: Speed component of offset axis

Fz : Speed component of a cylindrical interpolation linear axis before cutting point compensation

Fc : Speed component of cylindrical interpolation rotation axis before cutting point compensation
Vcs: Rotation axis component of a tool contact point vector (Vs in the figure) at the start point at a point in time
Vce: Rotation axis component of tool contact point vector (Ve in the figure) at the end point at a point in time
r : Radius of the cylinder of a rotary axis

Accordingly, the actual speed indication during circular interpolation is greater than the specified value when |Fc'| > |Fc|(inner offset of the arc). Conversely, the actual speed indication during circular interpolation is less than the specified value when |Fc'| < |Fc|(outer offset of the arc).

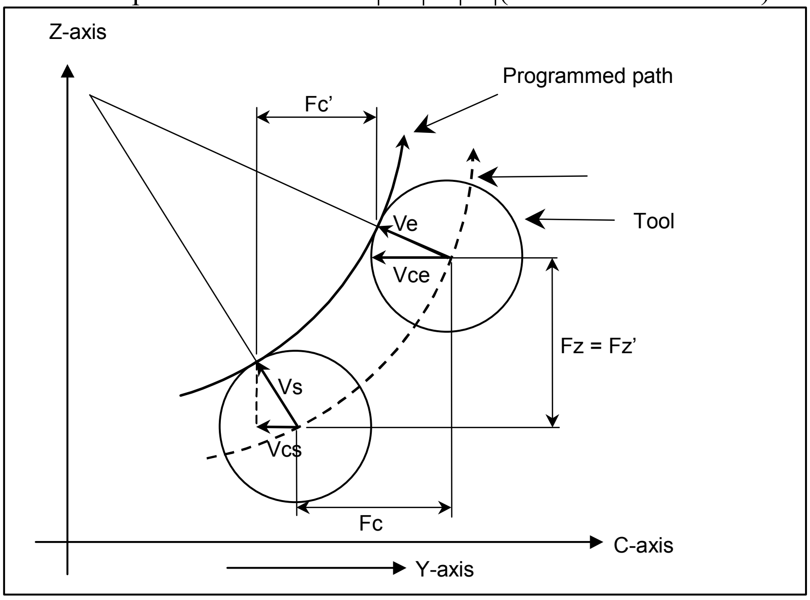


Fig.4.10 (l) Actual speed indication during circular interpolation

- Usable G codes

(1) In any of the following G code modes, cutting point interpolation for cylindrical interpolation can be specified:

G01,G02,G03	:	Linear interpolation, circular interpolation
G17,G18,G19	:	Plane selection
G22	:	Stored stroke check function on
G64	:	Cutting mode
G90,G91	:	Absolute command programming, incremental command programming
G94	:	Feed per minute

(2) Any of the following G codes can be specified in cutting point interpolation for cylindrical interpolation mode:

G01,G02 ,G03	:	Linear interpolation, circular interpolation
G04	:	Dwell
G40,G41,G42	:	Cutter compensation
G40.1 to G42.1	:	Normal direction control
G64	:	Cutting mode
G65 to G67	:	Macro call
G90,G91	:	Absolute command programming, incremental command programming

- Parameter

To enable this function, set bit 5 (CYA) of parameter No. 19530 to 1.

Limitation

- Overcutting during inner corner cutting

Theoretically, when the inner area of a corner is cut using linear interpolation as shown in Fig. 4.10(m), this function slightly overcuts the inner walls of the corner. This overcutting can be avoided by specifying a value of R that is slightly greater than the radius of the tool at the corner.

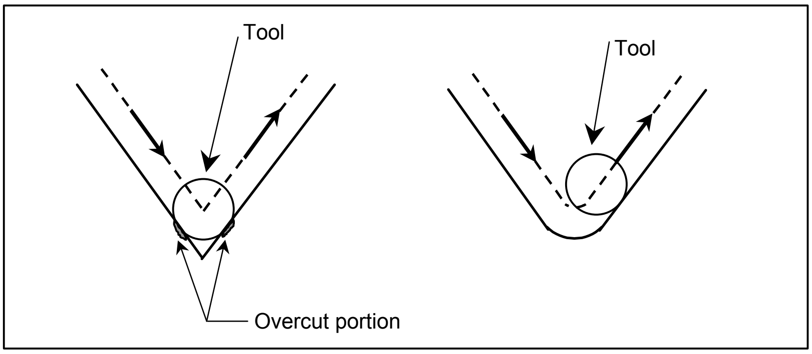


Fig.4.10 (m) Overcutting

- Setting the minimum input increment for an offset axis (Y-axis)

Set the same minimum input increment for an offset axis and linear axis when cylindrical interpolation is performed.

- Workpiece radius specification

When specifying the radius of a workpiece, use the minimum input increment (with no decimal point) for the linear axis used in cylindrical interpolation.

- Reference axis setting (parameter No. 1031)

When different increment systems are used for a linear axis and a rotary axis in cylindrical interpolation, specify the number of the linear axis in cylindrical interpolation to set the axis as the reference axis.

- Rotary axis roll-over function

When a rotary axis for which the rollover function is used is specified as a rotary axis to be used for cylindrical interpolation, the roll-over function is disabled in the cylindrical interpolation mode.
When the cylindrical interpolation is canceled, the rollover function is automatically enabled.

- Program restart

In a program restart operation, G07.1 for the cylindrical interpolation mode must not be specified.
Otherwise, PS0175 is issued.

Example

- Example of cutting point interpolation for cylindrical interpolation

The sample program below indicates the positional relationships between a workpiece and tool.

```
O0001 (CYLINDRICAL INTERPOLATION1) ;
N01 G00 G90 Z100.0 C0 ;
N02 G01 G91 G19 Z0 C0 ;
N03 G07.1 C57299 ;
N04 G01 G42 G90 Z120.0 D01 F250. ;        (1)
N05 C20.0 ;                               (2)
N06 G02 Z110.0 C60.0 R10.0 ;              (3)
N07 G01 Z100.0 ;                          (4)
N08 G03 Z60.0 C70.0 R40.0 ;               (5)
N09 G01 C100.0 ;
  :
M30 ;
```

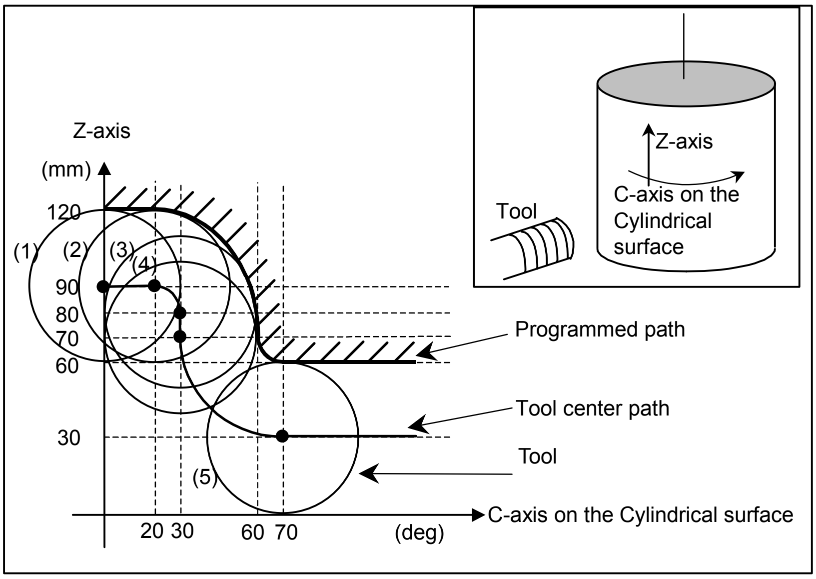


Fig. 4.10 (n) Path of sample program for cutting point interpolation for cylindrical interpolation

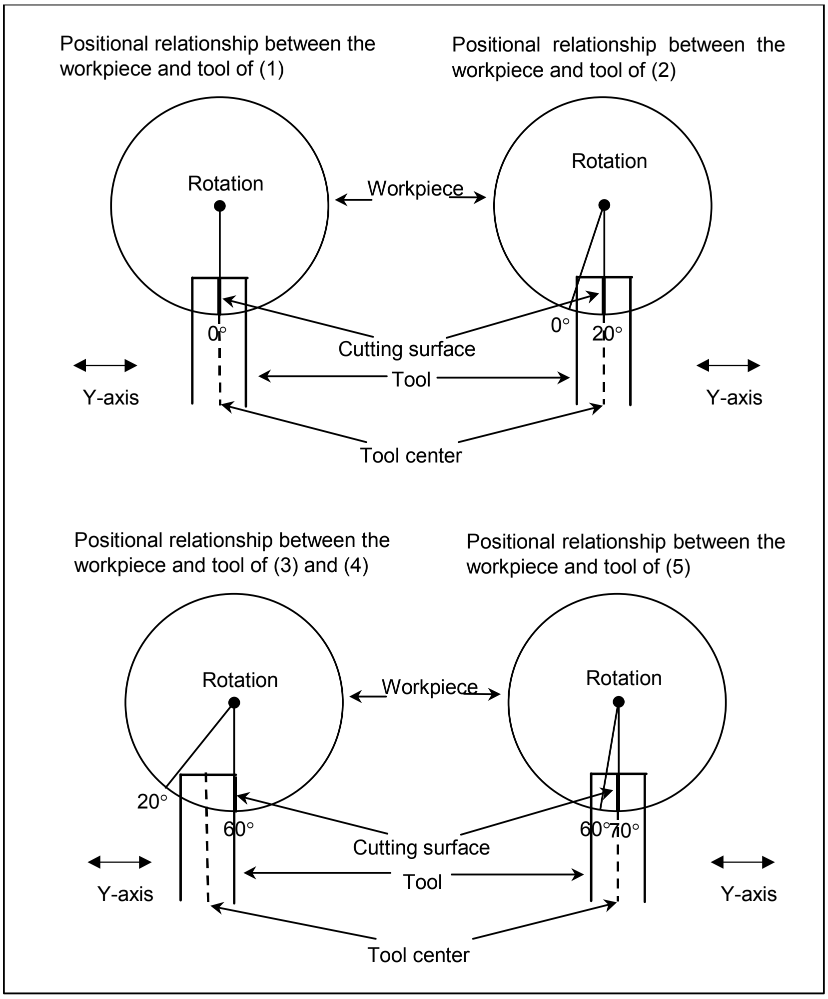


Fig. 4.10 (o) Positional relationships between workpiece and tool of sample program

The cutting surface in the rotary axis direction in (3) and (4) are uniform even if the cutter compensation amount is modified.

- Example of specifying cutting point interpolation for cylindrical interpolation and normal direction control at the same time

Cutter compensation No.01 is 30 mm.

```
O0002(CYLINDRICAL INTERPOLATION2) ;
N01 G00 G90 X100.0 A0 ;
N02 G01 G91 G17 X0 A0 ;
N03 G07.1 C57299 ;
N04 G01 G41 G42.1 G90 X120.0 D01 F250. ;
N05 A20.0 ;
N06 G03 X80.0 A60.0 R40.0 ;
N07 G01 X70.0 ;
N08 G02 X70.0 A70.0 R10.0 ;
N09 G01 A150.0 ;
N10 G02 X70.0 A190.0 R85.0 ;
N11 G01 X110.0 A265.0 ;
N12 G03 X120.0 A305.0 R85.0 ;
N13 G01 A360.0 ;
N14 G40 G40.1 X100.0 ;
N15 G07.1 A0 ;
N16 M30 ;
```

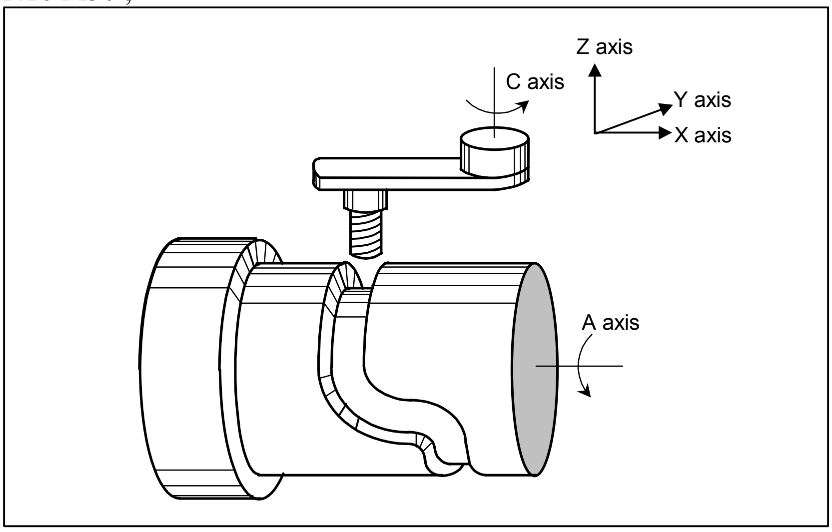
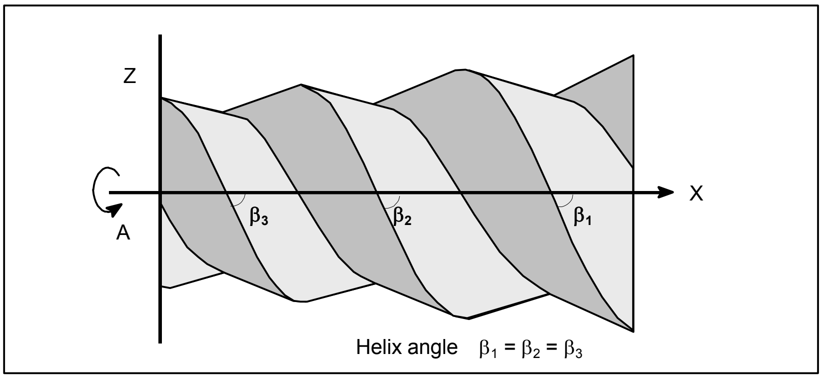


Fig. 4.10 (p) Example of specifying normal direction control at the same time

4.11 EXPONENTIAL INTERPOLATION (G02.3, G03.3)

Exponential interpolation exponentially changes the rotation of a workpiece with respect to movement on the rotary axis. Furthermore, exponential interpolation performs linear interpolation with respect to another axis. This enables tapered groove machining with a constant helix angle (constant helix taper machining). This function is best suited for grooving and grinding tools such as taper end mills.



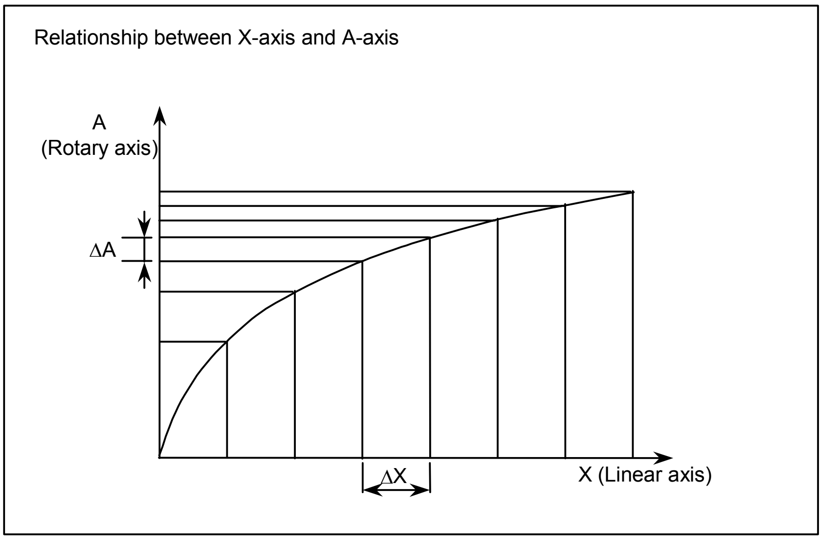


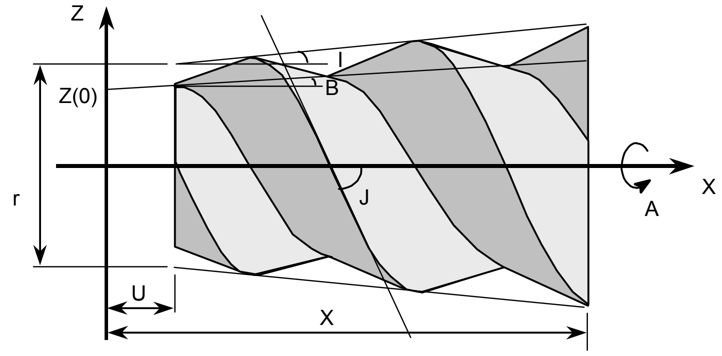
Fig. 4.11 (a) Exponential interpolation

Format

Positive rotation (ω = 0)
G02. 3 X_ Y_ Z_ I_ J_ K_ R_ F_ Q_ ;
Negative rotation (ω = 1)
G03. 3 X_ Y_ Z_ I_ J_ K_ R_ F_ Q_ ;

X_ : Specifies an end point with an absolute or incremental value.
Y_ : Specifies an end point with an absolute or incremental value.
Z_ : Specifies an end point with an absolute or incremental value.
I_ : Specifies angle I (from ±1 to ±89 deg in units of 0.001 deg).
J_ : Specifies angle J (from ±1 to ±89 deg in units of 0.001 deg).
K_ : Specifies the amount to divide the linear axis for exponential interpolation (span value). The specification unit depends on the reference axis. Specify a positive value.
The span value is specified in the manner specified in bit 0 (SPN) of parameter No. 5630. If SPN is set to 0, the division amount is specified in parameter No. 5643. If SPN is set to 1, the value specified with K becomes valid.
R_ : Specifies constant R for exponential interpolation. (See Explanations below.)
F_ : Specifies the initial feedrate.
Specified in the same way as an ordinary F code.
Specify a composite feedrate including a feedrate on the rotary axis.
Q_ : Specifies the feedrate at the end point.
The same unit used for F is used. The CNC internally performs interpolation between the initial feedrate (F) and final feedrate (Q), depending on the travel distance on the linear axis.

Explanation



r : Diameter of left end
U : Excess length
X : Amount of travel along the linear axis
I : Taper angle
B : Groove bottom taper angle
J : Helix angle

Fig. 4.11 (b) Constant helix machining for producing a tapered figure

In Fig. 4.11 (b) an absolute value on the X-axis, Z-axis, or A-axis is expressed as a function of workpiece rotation angle θ, such as *X*(θ), *Z*(θ), and *A*(θ).
Linear interpolation with the X-axis is performed for an axis other than the X-axis or A-axis. When *X*(θ) = 0, *A*(θ) = 0.

The relationship is expressed as follows.

$$Z(\theta) = \left\{ \frac{r}{2} - U * \tan(I) \right\} * (e^{\frac{\theta}{K}} - 1) * \frac{\tan(B)}{\tan(I)} + Z(0) \text{ .... (1)}$$

$$X(\theta) = \left\{ \frac{r}{2} - U * \tan(I) \right\} * (e^{\frac{\theta}{K}} - 1) * \frac{1}{\tan(I)} \text{ .... (2)}$$

$$A(\theta) = (-1)^{\omega} * \frac{360}{2\pi} * \theta$$

Where

$$K = \frac{\tan J}{\tan I}$$

ω: Helix direction (0: Positive, 1: Negative)

From Expressions (1) and (2), the following is obtained ;

$$Z(\theta) = \tan(B) * X(\theta) + Z(0) \text{ .... (3)}$$

From Expression (3), the Z-axis position is determined from a groove bottom taper angle (B) and X-axis position.
From Expression (1) and exponential definition expression (described later), the following is determined:

$$R = r/2 - U * \tan(I) \text{ .... (4)}$$

Constant R is determined from the left end diameter (r), excess length (U), and taper angle (I) according to Expression (4). Specify a taper angle (I) in address I, and specify a helix angle (J) in address J. Select a helix direction with G02.3 or G03.3.

- Exponential definition expressions

Exponential relational expressions for a linear axis and rotary axis are defined as follows:

$$X(\theta) = R * (e^{\frac{\theta}{K}} - 1) * \frac{1}{\tan(I)} \text{ .... (5)}$$

$$A(\theta) = (-1)^{\omega} * 360 * \frac{\theta}{2\pi} \text{ .... (6)}$$

Where,

$$K = \frac{\tan(J)}{\tan(I)}$$

$$\omega = 0/1$$

R, I, and J are constants, and θ represents an angle (radian).

- Span value K

A movement on an axis is carried out as linear interpolation in units of values obtained by dividing the movement on the X-axis by the span value (address K).
The following is obtained from Expression (5)

$$\theta(X) = K * \ln(\frac{X * \tan(I)}{R} + 1) \quad \text{.. (7)}$$

When there is movement from X_1 to X_2 on the linear axis, the amount of movement on the rotary axis is determined by:

$$\Delta\theta = K * \{\ln(\frac{X_2 * \tan(I)}{R} + 1) - \ln(\frac{X_1 * \tan(I)}{R} + 1)\}$$

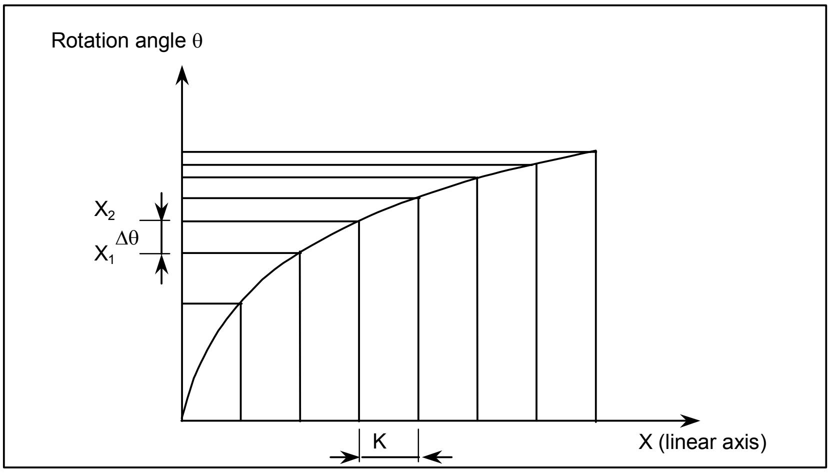


Fig. 4.11 (c) Span value K

- Rotation axis θ

In exponential interpolation, Expression <7> indicates the relationship between the X coordinate and the rotation angle θ about the A-axis. The expression in the parentheses of the natural logarithm ln in Expression <7> must satisfy Expression <8> indicated below, because of the condition of ln (the value in the parentheses is positive).

$$\frac{X * \tan(I)}{R} > -1 \quad \text{.......} \quad (8)$$

If the value of X*tan(I)/R becomes -1 or less, the position comes to the right of point (A) in Fig. 4.11 (d). Because the resultant figure is unfeasible, an alarm PS5062 will be issued.

In exponential interpolation, the X, Y, Z, and U values are handled as coordinates in the workpiece coordinate system. If a positive value specified in incremental programming corresponds to a negative value in the workpiece coordinate system, the negative value is used in the calculation.

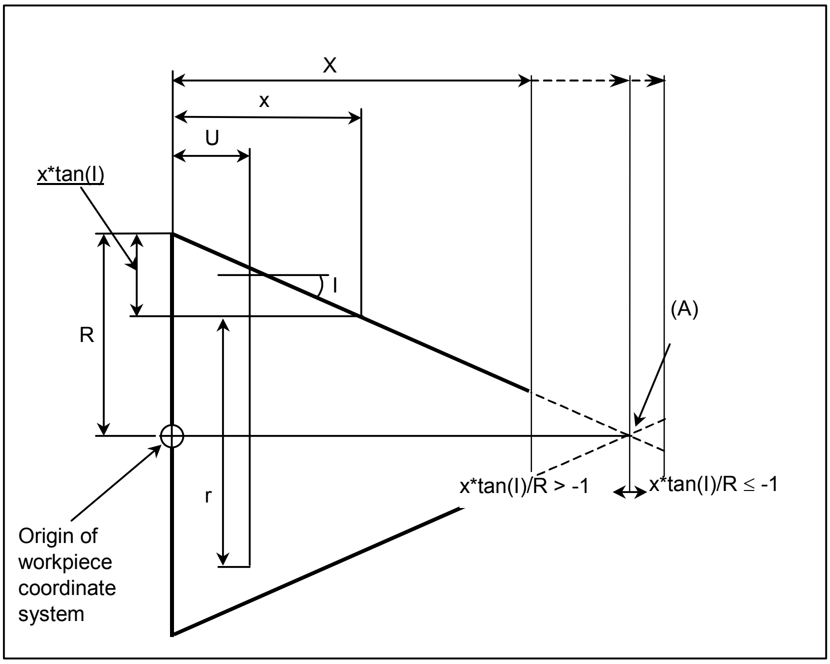


Fig. 4.11 (d) Rotation angle θ

- Taper angle I

The machining profile and the sign of taper angle I have the following relationships:

- If the profile tapers up toward the right, the I value is positive.
- If the profile tapers down toward the right, the I value is negative.

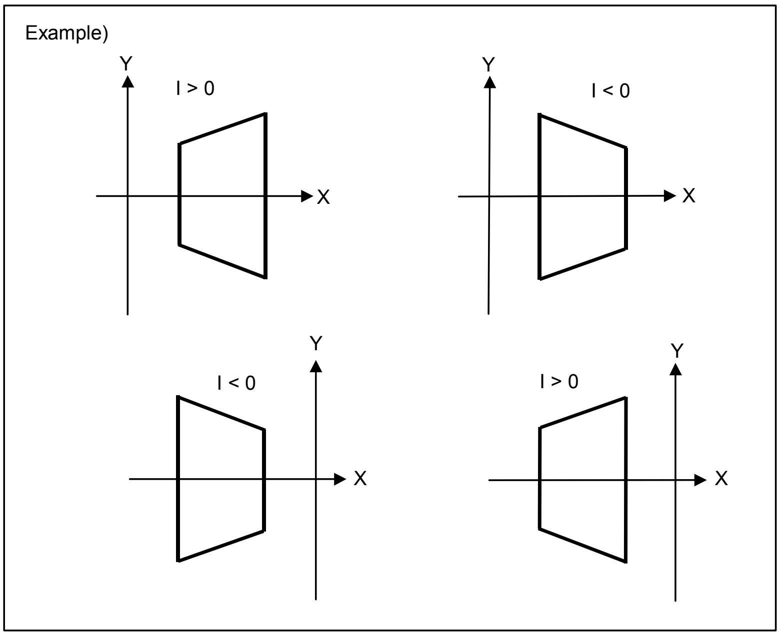


Fig. 4.11 (e) Taper angle I

- Helix angle J

The sign of the helix angle J is assigned as illustrated below.

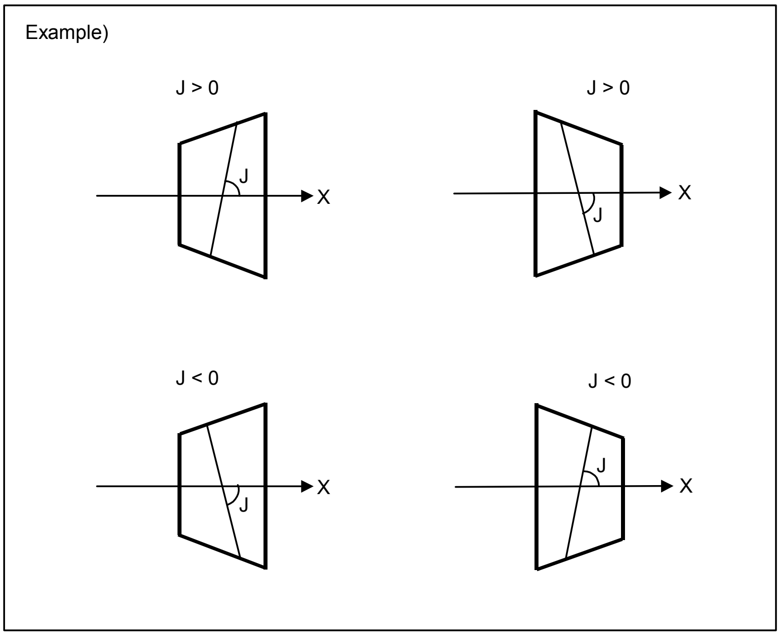
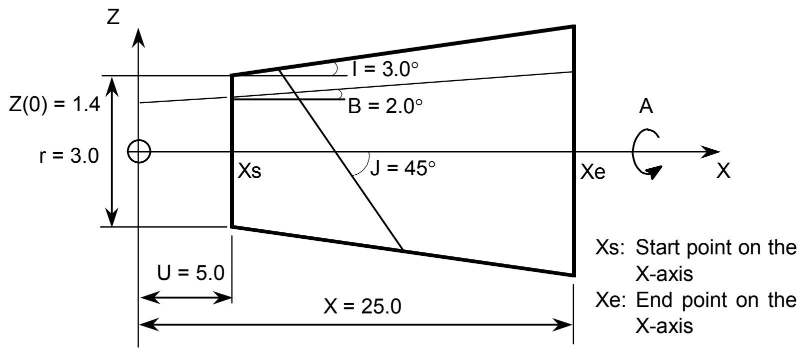


Fig. 4.11 (f) Helix angle J

Example



The start point and end point of the Z-axis and the constant R are obtained from the following expressions:

Start point on the Z-axis $= \tan(B) * Xs + Z(0)$

End point on the Z-axis $= \tan(B) * Xe + Z(0)$

$R = r/2 - U * \tan(I)$

Limitation

- Cases where linear interpolation is performed

Even when the G02.3 or G03.3 mode is set, linear interpolation is performed in the following cases:

- When the linear axis specified in parameter (No. 5641) is not specified, or the amount of movement on the linear axis is 0
- When the rotary axis specified in parameter (No. 5642) is specified
- When the amount for dividing the linear axis (span value) is 0

- Tool compensation

The tool compensation functions (tool length compensation, cutter or tool nose radius compensation, and three-dimensional tool compensation) cannot be used in the G02.3 or G03.3 mode.

> ⚠ **CAUTION**
> The amount for dividing the linear axis for exponential interpolation (span value) affects figure precision. However, if an excessively small value is set, the machine may stop during interpolation. Try to specify an optimal span value depending on the machine being used.

4.12 SMOOTH INTERPOLATION (G05.1)

Either of two types of machining can be selected, depending on the program command.

- For those portions where the accuracy of the figure is critical, such as at corners, machining is performed exactly as specified by the program command.
- For those portions having a large radius of curvature where a smooth figure must be created, points along the machining path are interpolated with a smooth curve, calculated from the polygonal lines specified with the program command (smooth interpolation).

In this manner, high-speed, high-precision machining can be performed.

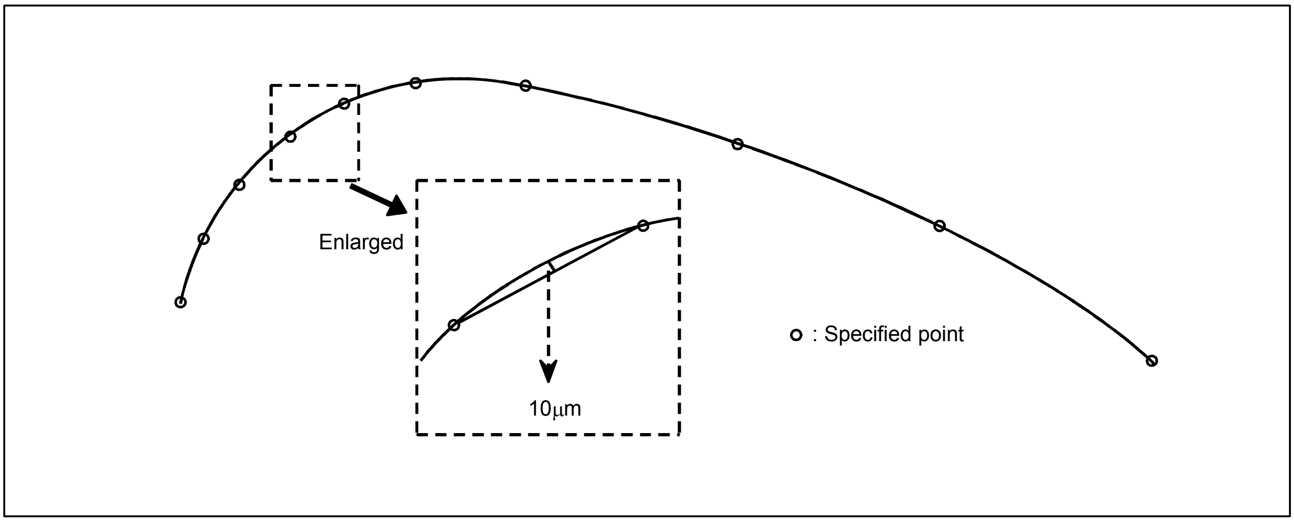
Format

G05.1Q2X0Y0Z0 ; Starting of smooth interpolation mode
:
G05.1Q0 ; Cancellation of smooth interpolation mode

Explanation

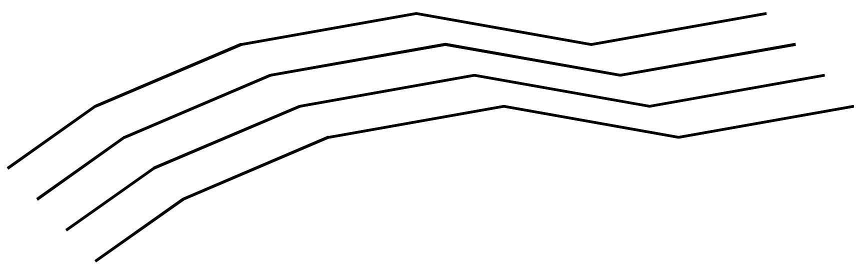
- Characteristics of smooth interpolation

To machine a part having sculptured surfaces, such as metal moldings used in automobiles and airplanes, a part program usually approximates the sculptured surfaces with minute line segments. As shown in the following figure, a sculptured curve is normally approximated using line segments with a tolerance of about 10 μm.



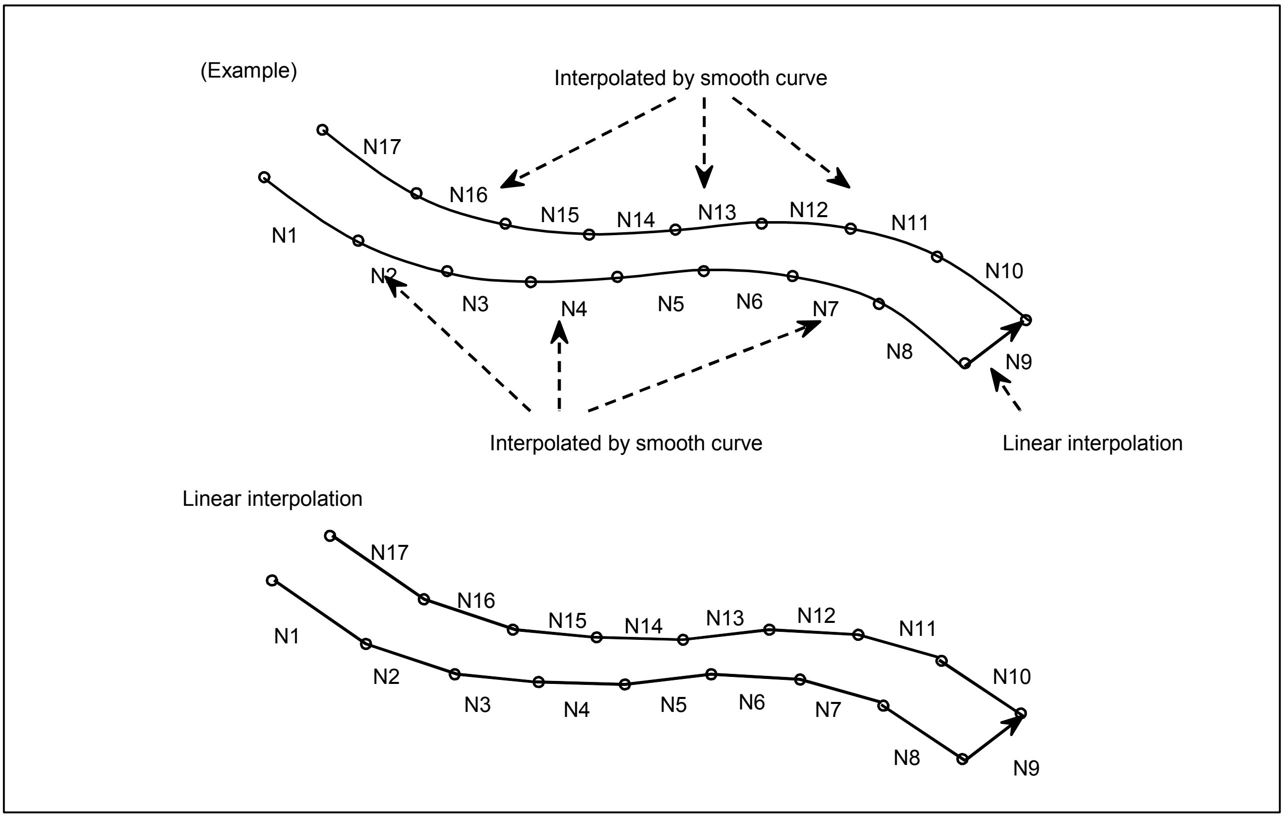
When a program approximates a sculptured curve with line segments, the length of each segment differs between those portions that have mainly a small radius of curvature and those that have mainly a large radius of curvature. The length of the line segments is short in those portions having a small radius of curvature, while it is long in those portions having a large radius of curvature. Because linear interpolation controls the tool movement exactly as programmed so as not to let the tool depart from the path specified by a part program, machining is performed exactly along line segments with which a sculptured curve is approximated. Consequently, when a curve with a large radius of curvature and a moderate curvature change is machined, the corners of line segments may become apparent. Such concave and convex portions, produced as a result of machining performed exactly as specified, are found troublesome when a smooth surface must be obtained by finishing.

Profile	Portions having mainly a small radius of curvature	Portions having mainly a large radius of curvature
Example of machined parts	Automobile parts	Decorative parts, such as body side moldings
Length of line segment	Short	Long
Resulting surfaces produced using high-precision contour control	Smooth surface even when machining is performed exactly as specified by a program	Uneven surfaces may result when machining is performed exactly as specified by a program



Example of uneven surfaces (polygon) resulting from machining that precisely follows the line segments.

In smooth interpolation mode, the CNC automatically determines, according to the program command, whether an accurate figure is required, such as at corners, or a smooth figure is required where the radius of curvature is large. If a block specifies a travel distance or direction which differs greatly from that in the preceding block, smooth interpolation is not performed for that block. Linear interpolation is performed exactly as specified by the program command. Programming is thus very simple.



- Conditions for performing smooth interpolation

Smooth interpolation is performed when all the following conditions are satisfied. If any of the following conditions is not satisfied for a block, that block is executed without smooth interpolation then the conditions are checked for the next block.

(1) The machining length specified in the block is shorter than the length specified with parameter No. 8486.

(2) The machining length is other than 0.

(3) The modes are:

G01	:	Linear interpolation
G13.1	:	Polar coordinate interpolation cancel
G15	:	Polar coordinate command cancel
G40	:	Cutter or tool nose radius compensation cancel (except for 3-dimensional tool compensation)
G64	:	Cutting mode
G80	:	Canned cycle cancel
G94	:	Feed per minute

(4) Machining is specified only along the axes specified with G05.1Q2.

(5) The block is judged to be unsuitable for smooth interpolation, as performed with the internal algorithm of the CNC.

- Commands which cancel smooth interpolation

(1) Auxiliary and second auxiliary functions

(2) M98, M99 : Subprogram call
M198 : Calling a subprogram in external memory

Limitation

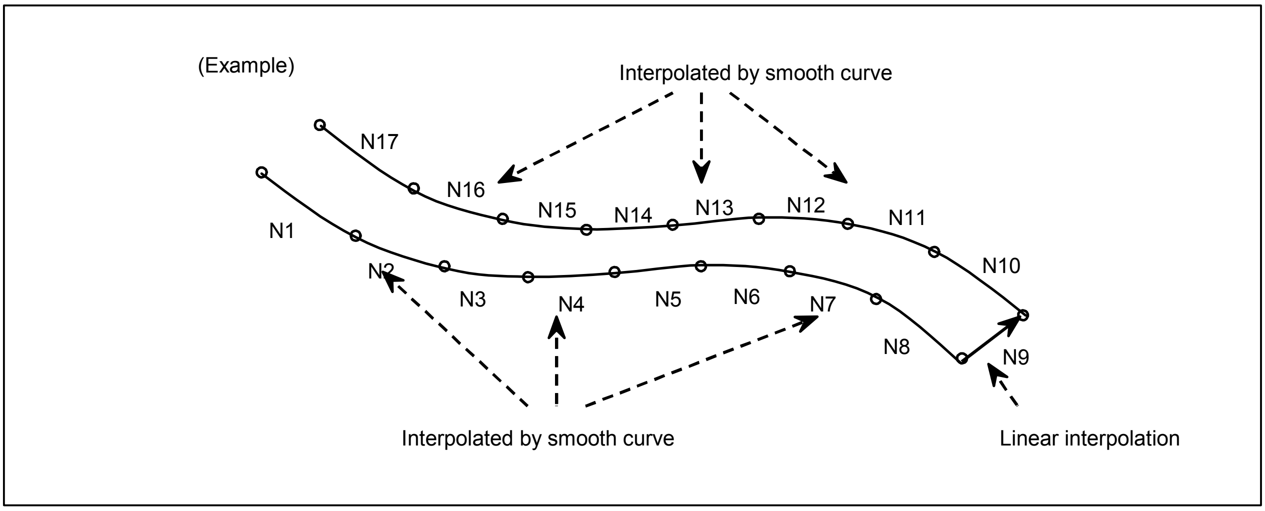
- Controlled axes

Smooth interpolation can be specified only for the X-, Y-, and Z-axes and any axes parallel to these axes (up to three axes at one time).

Example

<Example program for smooth interpolation>

```
 .
 .
G91 ;
G05. 1  Q2  X0  Y0  Z0 ;
N01  G01  X1000  Z-300 ;
N02  X1000  Z-200 ;
N03  X1000  Z-50 ;
N04  X1000  Z50 ;
N05  X1000  Z50 ;
N06  X1000  Z-25 ;
N07  X1000  Z-175 ;
N08  X1000  Z-350 ;
N09  Y1000 ;
N10  X-1000  Z350 ;
N11  X-1000  Z175 ;
N12  X-1000  Z25 ;
N13  X-1000  Z- 50 ;
N14  X-1000  Z- 50 ;
N15  X-1000  Z50 ;
N16  X-1000  Z200 ;
N17  X-1000  Z300 ;
G05. 1  Q0 ;
 .
 .
```



4.13 NANO SMOOTHING

Overview

When a desired sculptured surface is approximated by minute segments, the nano smoothing function generates a smooth curve inferred from the programmed segments and performs necessary interpolation.
The nano smoothing function infers a curve from a programmed figure approximated with segments within tolerance. If the spacing between adjacent inflection points or programmed points is not constant, this function can generate a smoother curve than the conventional smooth interpolation function.
The interpolation of the curve reduces the segment approximation error, and the nano-interpolation makes the cutting surface smoother.
This function needs the AI contour control option.

Format

G5.1 Q3 Xp0 Yp0 Zp0 ; : Nano smoothing mode on
G5.1 Q0 ; : Nano smoothing mode off
Xp: X-axis or an axis parallel to the X-axis
Yp: Y-axis or an axis parallel to the Y-axis
Zp: Z-axis or an axis parallel to the Z-axis

NOTE
1. Specify G5.1 alone in a block. (Avoid specifying any other G code in the same block.)
2. Specify position 0 for the axis programmed in the nano smoothing mode on block. The specified axis is subjected to nano smoothing, but no movement is made even in the absolute programming mode.

- Nano smoothing mode

Specifying G5.1 Q3 selects the nano smoothing mode. An axis of nano smoothing is specified in the same block. The three basic axes (X, Y, and Z) and their parallel axes can be specified as the axis of nano smoothing. In the nano smoothing mode, nano smoothing is performed in a block which satisfies the conditions described below. Nano smoothing is cancelled in a block which does not satisfy the conditions.
Specifying G5.1 Q3 also enables AI contour control. The automatic velocity control by AI contour control reduces impacts on the mechanical system.
Specifying G5.1 Q0 cancels the nano smoothing mode. The AI contour control mode is also cancelled at the same time.
Making a reset also cancels the nano smoothing mode.

- Conditions to enable nano smoothing

Nano smoothing is enabled when the following conditions are satisfied. Nano smoothing is cancelled in a block which does not satisfy the conditions. A decision is made to perform nano smoothing from the next block.

<1> The programmed block length is shorter than the length specified in parameter No. 8486.
<2> The programmed block length is longer than the length specified in parameter No. 8490.
<3> The difference in angle between adjacent programmed blocks is smaller than the value specified in parameter No. 8487.
<4> The current mode is one of the following:

- Linear interpolation
- Feed per minute
- Cutter compensation cancel
- Canned cycle cancel
- Scaling cancel
- Macro modal call cancel
- Constant surface speed control cancel
- Cutting mode
- Coordinate system rotation/three-dimensional coordinate conversion cancel
- Polar coordinate cancel
- Normal-direction control cancel
- Polar coordinate interpolation cancel
- Programmable mirror image cancel
- Polygon turning cancel

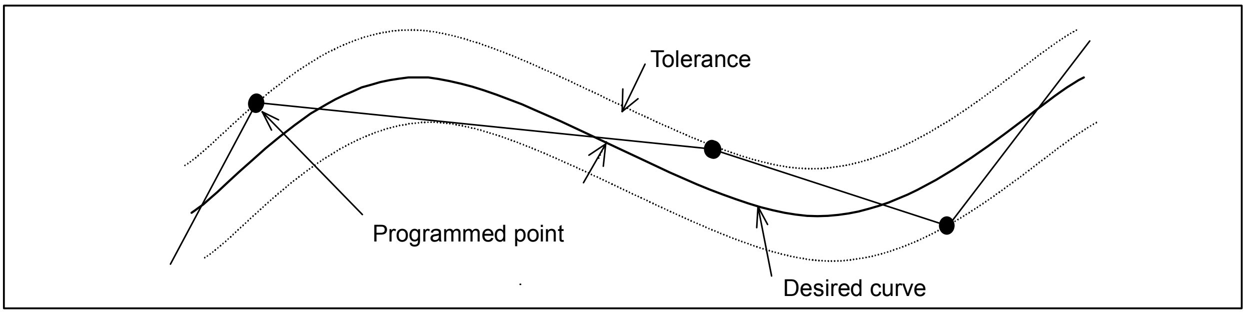
<5> No one-shot G code is specified in the block.
<6> Buffering is not suppressed in the block.
<7> A movement just on the axis of nano smoothing is specified in the block.

- Checking the nano smoothing mode

Diagnostic data (No. 5000) indicates whether the nano smoothing mode is enabled in the current block.
If the nano smoothing mode is enabled, the "smoothing on" bit is set to 1.

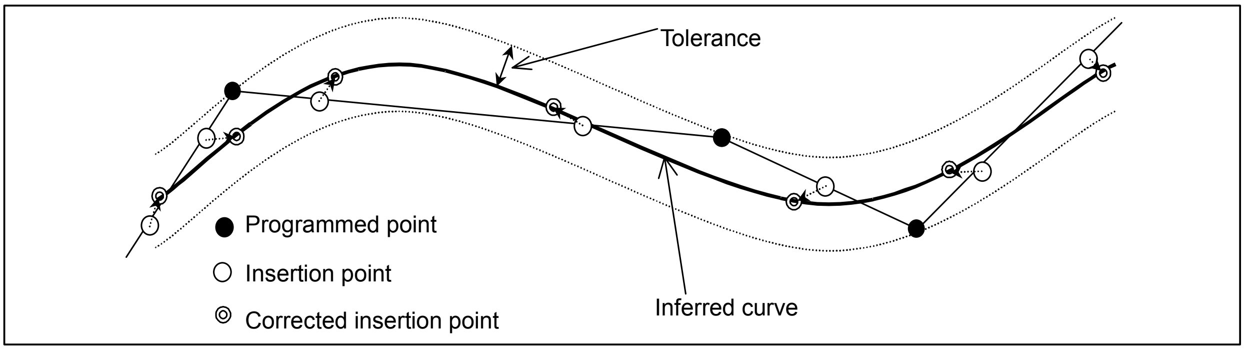
Explanation

Generally, a program approximates a sculptured surface with minute segments with a tolerance of about 10 μm.



Many programmed points are placed on the boundary of tolerance. The programmed points also have a rounding error owing to the least input increment of the CNC. The nano smoothing function creates multiple insertion points between adjacent programmed points so that a smooth curve can be created from the approximation segments. The desired curve is inferred from the insertion points of multiple blocks including buffered blocks.

Many insertion points are closer to the desired curve than the programmed points. A stable curve can be inferred with the insertion points created from multiple blocks including buffered blocks. Because the position of each insertion point is corrected in a unit smaller than the least input increment of the CNC within tolerance, the impact of rounding error is reduced.
Nano-interpolation is performed for the curve inferred from the corrected insertion points, so the resultant cutting surface becomes smooth.



- Specifying the tolerance

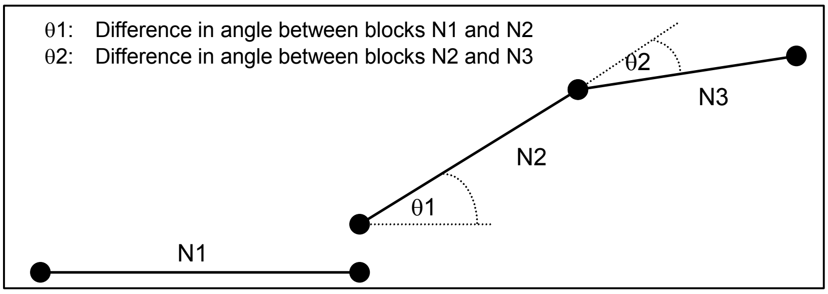
The tolerance of the program of nano smoothing is specified in parameter No. 19581.
The insertion points are corrected within tolerance, and a curve is inferred accordingly.
If 0 is specified in parameter No. 19581, the minimum travel distance in the increment system is considered to be the tolerance.

- Making a decision on the basis of the spacing between adjacent programmed points

If the spacing between adjacent programmed points (block length) exceeds the value specified in parameter No. 8486 or falls below the value specified in parameter No. 8490 in the nano smoothing mode, the nano smoothing mode is cancelled at the start point of the block. Linear interpolation can be performed in the block.
If the values specified in the parameters are 0, no decision is made on the basis of the spacing between adjacent programmed points.

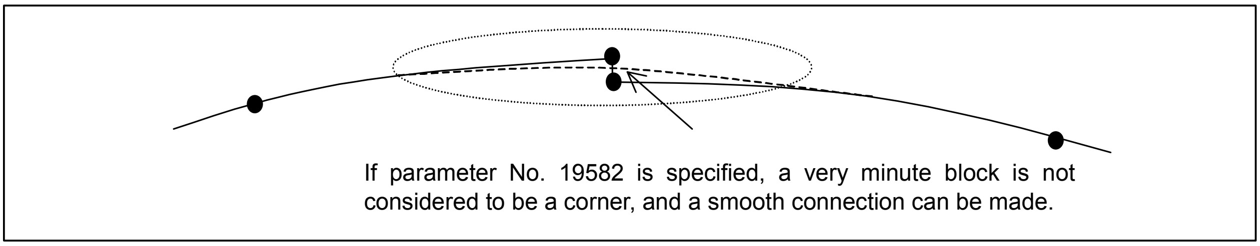
- Making a decision at a corner

If the difference in angle (see the following figure) between adjacent programmed blocks exceeds the value specified in parameter No. 8487 in the nano smoothing mode, the nano smoothing mode is cancelled at the corner.



If the value specified in the parameter is 0, no decision is made at the corner on the basis of the difference in angle.
Very minute blocks created for some reasons such as a calculation error of CAM can be ignored, and a smooth connection can be made at a corner. To do this, specify parameter No. 19582 to the minimum travel distance with which a decision is made on the basis of difference in angle. Then, the decision at a corner is disabled for a block of which distance is less than the specified minimum travel distance.

However, a decision based on the spacing between adjacent programmed points specified in parameter No. 8490 has higher priority than the decision at a corner. Therefore, the value specified in parameter No. 19582 must be greater than the value specified in parameter No. 8490.



Limitation

- Single-block operation

When single-block operation is carried out in the nano smoothing mode, the operation stops at a corrected insertion point not at a programmed point.
Even in the nano smoothing mode, normal single-block operation is carried out for a block that does not satisfy the conditions of nano smoothing mode.

- Tool length compensation

To carry out tool length compensation, specify the command before specifying nano smoothing. Avoid changing the amount of compensation in the nano smoothing mode.
If G43, G44, or G49 is specified in a block between the block in which the command of nano smoothing mode on (G5.1 Q3) is specified and the block in which the command of nano smoothing mode off (G5.1 Q0) is specified, an alarm PS0343 will be issued.

- Cutter or tool nose radius compensation

If cutter or tool nose radius compensation is specified in the nano smoothing mode, the nano smoothing mode is cancelled. Then, when the command of cutter or tool nose radius compensation cancel (G40) is specified, a decision is made whether to start nano smoothing from the next block. The startup and cancel operations of type C are always carried out for the cutter or tool nose radius compensation specified in the nano smoothing mode, irrespective of the parameter setting.
A command related to cutter or tool nose radius compensation should not be specified in the nano smoothing mode unless it is absolutely necessary.

- Interruption type custom macro

No interruption type custom macro can be used in the nano smoothing mode.
If the nano smoothing mode is specified while an interruption type custom macro is enabled or if an interruption type custom macro is enabled in the nano smoothing mode, an alarm PS0342 will be issued.

- Manual intervention

Manual intervention by specifying the manual absolute on command cannot be performed in the nano smoothing mode. If this is attempted, an alarm PS0340 will be issued at the cycle start after manual intervention.

- Rotary table dynamic fixture offset

The command of rotary table dynamic fixture offset (G54.2) must be cancelled before specifying the nano smoothing mode. These commands cannot be used in the nano smoothing mode. If an attempt is made to use one of these commands, an alarm PS0343 will be issued.

- Binary operation by remote buffer

Binary operation by remote buffer cannot be carried out in the nano smoothing mode.
To carry out the binary operation, cancel the nano smoothing mode in advance.

- Number of blocks that can be specified successively

Up to about 300,000,000 blocks can be specified successively in the nano smoothing mode. If more blocks are specified, an alarm PS0341 will be issued.
However, when a block which does not satisfy the conditions of the nano smoothing mode is encountered, the mode is canceled and the counted number of successive blocks is reset to 0.

- Continuity of a program

Curve interpolation is carried out for multiple programmed blocks including buffered blocks in the nano smoothing mode.
Therefore, the programmed commands must be executed continuously in the nano smoothing mode.
The continuity of a program may be lost, and continuous execution may not be performed, in some cases such as the following: A single-block stop is made in the nano smoothing mode; and another program is executed in the MDI mode. If this occurs, an alarm PS0344 will be issued.

- Restrictions on resumption of automatic operation

(1) Program restart
Curve interpolation is performed for corrected insertion points not for programmed points in the nano smoothing mode. Accordingly, when a sequence number is specified to restart the program, the operation cannot be restart from a programmed point in a block.
To restart a program, specify a block number, using the block counter displayed in the program screen.

(2) Block restart
A block cannot be restarted in the nano smoothing mode. When an attempt is made to restart a block, an alarm PS0344 will be issued.

(3) Tool retract and recover
The tool cannot be retracted or recovered in the nano smoothing mode.

(4) Retracing
Retracing cannot be performed in the nano smoothing mode.

(5) Active block cancel
The active block cancel function is temporarily disabled in the nano smoothing mode.

- Functions that cannot be used simultaneously

The nano smoothing function cannot be used simultaneously with the following functions.

- Parallel axis control
- Twin table control

4.14 NURBS INTERPOLATION (G06.2)

Many computer-aided design (CAD) systems used to design metal dies for automobiles and airplanes utilize non-uniform rational B-spline (NURBS) to express a sculptured surface or curve for the metal dies.

This function enables NURBS curve expression to be directly specified to the CNC. This eliminates the need for approximating the NURBS curve with minute line segments. This offers the following advantages:

1. No error due to approximation of a NURBS curve by small line segments
2. Short part program
3. No break between blocks when small blocks are executed at high speed
4. No need for high-speed transfer from the host computer to the CNC

When this function is used, a computer-aided machining (CAM) system creates a NURBS curve according to the NURBS expression output from the CAD system, after compensating for the length of the tool holder, tool diameter, and other tool elements. The NURBS curve is programmed in the NC format by using these three defining parameters: control point, weight, and knot.

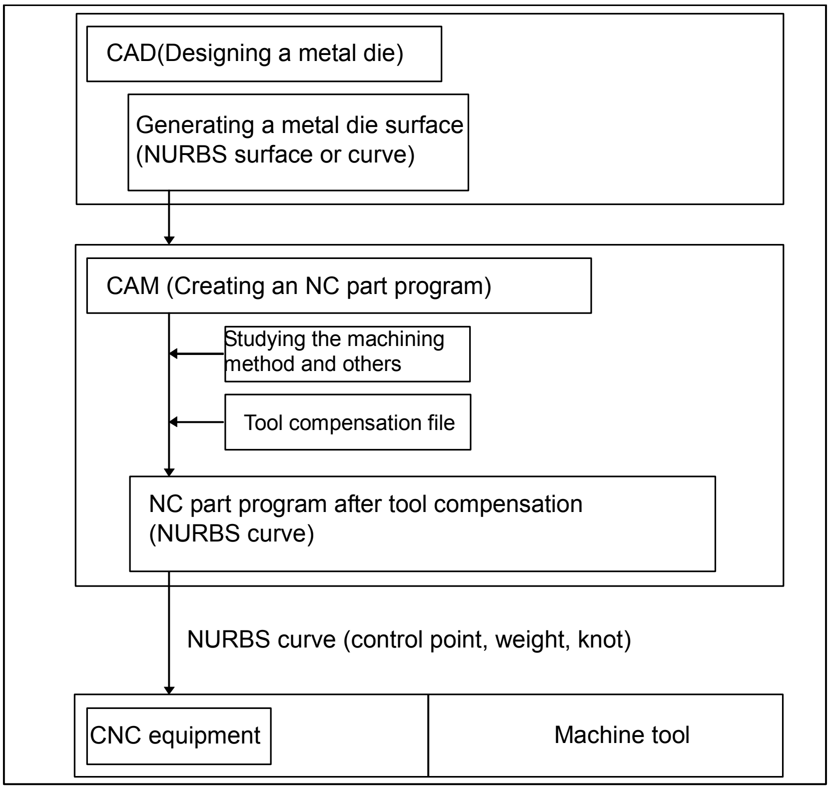


Fig. 4.14 (a) NC part program for machining a metal die according to a NURBS curve

Format

```
G06.2[P_]K_X_Y_Z_[R_] [F_];
      K_X_Y_Z_[R_];
      K_X_Y_Z_[R_];
      K_X_Y_Z_[R_];
       :
      K_X_Y_Z_[R_];
      K_;
      :
      K_;
G01 . . .

G06.2  : Start NURBS interpolation mode
P_     : Rank of NURBS curve
X_Y_Z_ : Control point
R_     : Weight
K_     : Knot
F_     : Feedrate
```

Explanation

- NURBS interpolation mode

NURBS interpolation mode is selected when G06.2 is programmed. G06.2 is a modal G code of group 01. NURBS interpolation mode ends when a G code of group 01 other than G06.2 (G00, G01, G02, G03, etc.) is specified.

- Rank of NURBS

A rank of NURBS can be specified with address P. The rank setting, if any, must be specified in the first block. If the rank setting is omitted, a rank of four (degree of three) is assumed for NURBS. The valid data range for P is 2 to 4. The P values have the following meanings:

P2: NURBS having a rank of two (degree of one)
P3: NURBS having a rank of three (degree of two)
P4: NURBS having a rank of four (degree of three) (default)

This rank is represented by k in the defining expression indicated in the description of NURBS curve below. For example, a NURBS curve having a rank of four has a degree of three. The NURBS curve can be expressed by the constants t^3, t^2, and t^1.

- Weight

The weight of a control point programmed in a single block can be defined. When the weight setting is omitted, a weight of 1.0 is assumed.

- Knot

The number of specified knots must equal the number of control points plus the rank value. In the blocks specifying the first to last control points, each control point and a knot are specified in an identical block. After these blocks, as many blocks (including only a knot) as the rank value are specified. The NURBS curve programmed for NURBS interpolation must start from the first control point and end at the last control point. The first k knots (where k is the rank) must have the same values as the last k knots (multiple knots). If the absolute coordinates of the start point of NURBS interpolation do not match the position of the first control point, alarm PS5117 is issued. (To specify incremental values, G06.2 X0 Y0 Z0 K_ must be programmed.)

- NURBS curve

Using these variables:

k : Rank
Pi : Control point
Wi : Weight
Xi : Knot ($X_i < X_{i+1}$)
Knot vector $[X_0, X_1, \ldots, X_m]$ ($m = n+ k$)
t : Spline parameter,

the spline basis function N can be expressed with the de Boor-Cox recursive formula, as indicated below:

$$N_{i,1}(t) = \begin{cases} 1(x_i < t < 1x_{i+1}) \\ 0(t < x_i, x_{i+1} < t) \end{cases}$$

$$N_{i,k}(t) = \frac{(t - x_i)N_{i,k-1}(t)}{x_{i+k-1} - x_i} + \frac{(x_{i+k} - t)N_{i+1,k-1}(t)}{x_{i+k} - x_{i+1}}$$

The NURBS curve P(t) of interpolation can be expressed as follows:

$$P(t) = \frac{\sum_{i=0}^{n} N_{i,k}(t) w_i p_i}{\sum_{i=0}^{n} N_{i,k}(t) w_1}$$

($X0 \le t \le Xm$)

- Reset

A reset during NURBS interpolation results in the clear state. The modal code of group 1 enters the state specified in the parameter G01 (No. 3402#0).

Limitation

- Controlled axes

NURBS interpolation can be performed on up to three axes. The axes of NURBS interpolation must be specified in the first block. A new axis cannot be specified before the beginning of the next NURBS curve or before NURBS interpolation mode ends.

- Command in NURBS interpolation mode

In NURBS interpolation mode, any command other than the NURBS interpolation command (miscellaneous function and others) cannot be specified.

- Manual intervention

If manual intervention is attempted while manual absolute mode is set, alarm PS5118 is issued.

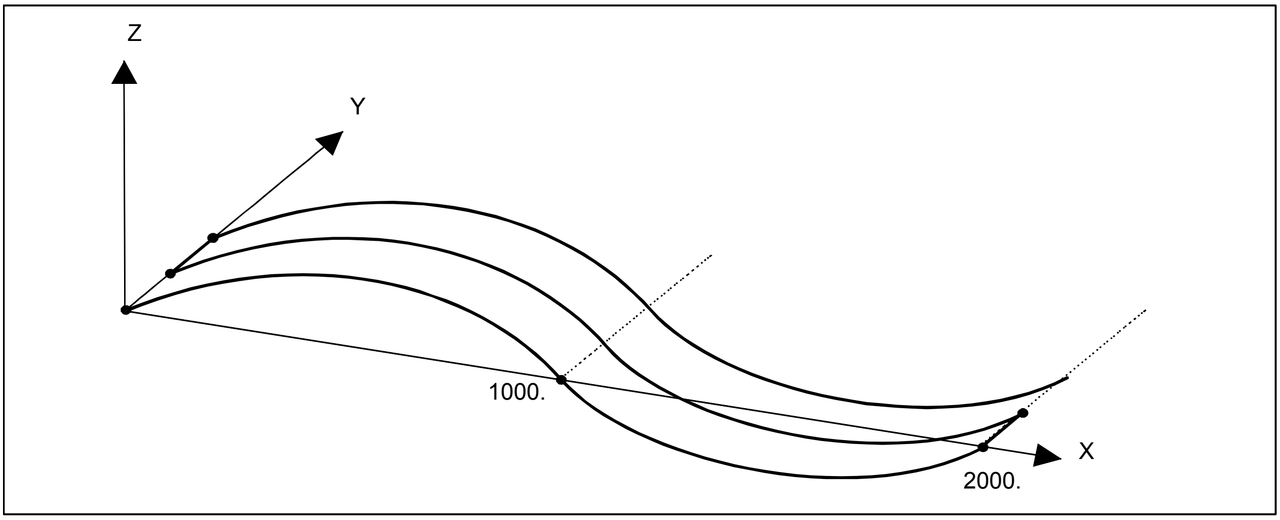
- Cutter compensation

Cutter compensation cannot be simultaneously executed. NURBS interpolation can only be specified after cutter compensation has been canceled.

Example

<Sample NURBS interpolation program>

```
G90;
...
G06.2   K0.   X0.      Z0.;
        K0.   X300.    Z100.;
        K0.   X700.    Z100.;
        K0.   X1300.   Z-100.;
        K0.5  X1700.   Z-100.;
        K0.5  X2000.   Z0.;
        K1.0;
        K1.0;
        K1.0;
        K1.0;
G01     Y0.5;
G06.2   K0.   X2000.   Z0.;
        K0.   X1700.   Z-100.;
        K0.   X1300.   Z-100.;
        K0.   X700.    Z100.;
        K0.5  X300.    Z100.;
        K0.5  X0.      Z0.;
        K1.0;
        K1.0;
        K1.0;
        K1.0;
G01     Y0.5;
G06.2 ...
...
G01 ...
```



4.15 HYPOTHETICAL AXIS INTERPOLATION (G07)

In helical interpolation, when pulses are distributed with one of the circular interpolation axes set to a hypothetical axis, sine interpolation is enabled.
When one of the circular interpolation axes is set to a hypothetical axis, pulse distribution causes the speed of movement along the remaining axis to change sinusoidally. If the major axis for threading (the axis along which the machine travels the longest distance) is set to a hypothetical axis, threading with a fractional lead is enabled. The axis to be set as the hypothetical axis is specified with G07.

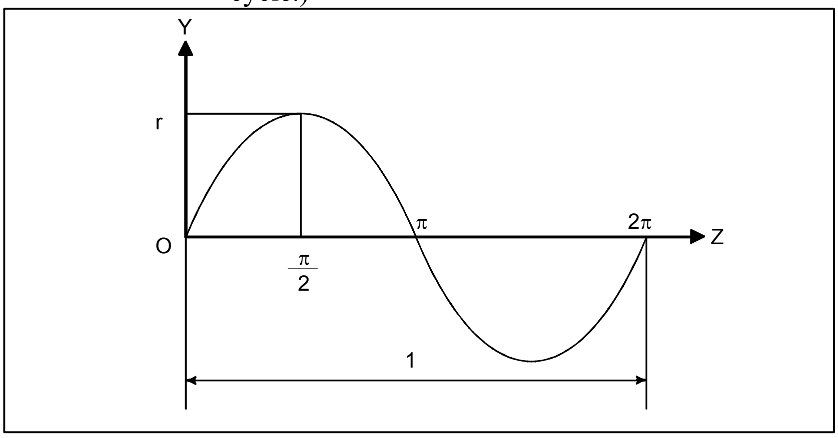
Format

G07 α0 ; : Hypothetical axis setting
:
G07 α1 ; : Hypothetical axis cancel
Where, α is any one of the addresses of the controlled axes.

Explanation

- Sine interpolation

The a axis is regarded as a hypothetical axis for the period of time from the G07 α0 command until the G07 α1 command appears.
Suppose sine interpolation is performed for one cycle in the YZ plane. The hypothetical axis is then the X axis.
$X^2+Y^2 = r^2$ (r is the radius of an arc.)
$Y = r\mathrm{SIN}\,(\frac{2\pi}{1}Z)$ (1 is the distance traveled along the Z-axis in one cycle.)



- Interlock, stroke limit, and external deceleration

Interlock, stroke limit, and external deceleration can also apply to the hypothetical axis.

- Handle interruption

An interrupt caused by the handle also applies to the hypothetical axis. This means that movement for a handle interrupt is performed.

Limitation

- Manual operation

The hypothetical axis can be used only in automatic operation. In manual operation, it is not used, and movement takes place.

- Move command

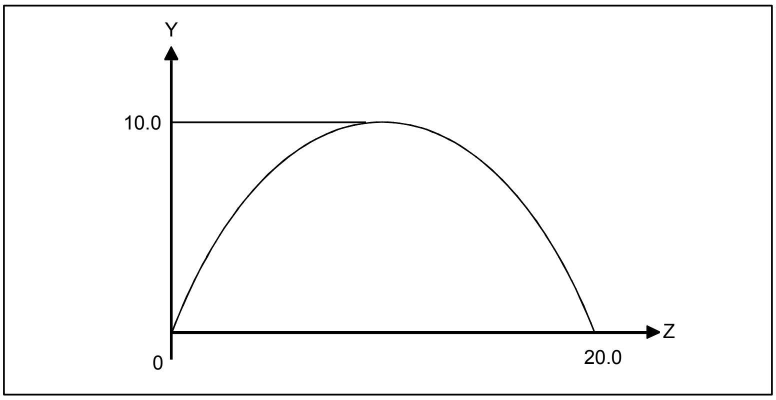
Specify hypothetical axis interpolation only in the incremental mode.

- Coordinate rotation

Hypothetical axis interpolation does not support coordinate rotation.

Example

- Sine interpolation



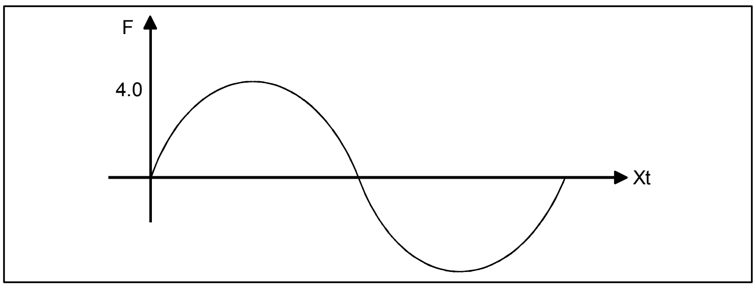
N001 G07 X0 ;
N002 G91 G17 G03 X-20.0 Y0.0 I-10.0 Z20.0 F100 ;
N003 G01 X10.0 ;
N004 G07 X1 ;

From the N002 to N003 blocks, the X-axis is set to a hypothetical axis. The N002 block specifies helical cutting in which the Z-axis is the linear axis. Since no movement takes place along the X axis, movement along the Y-axis is performed while performing sine interpolation along the Z-axis.
In the N003 block, there is no movement along the X-axis, and so the machine dwells until interpolation terminates.

- Changing the feedrate to form a sine curve

(Sample program)

G07Z0 ;	The Z-axis is set to a hypothetical axis.
G02X0Z0I10.0F4. ;	The feedrate on the X-axis changes sinusoidally.
G07Z1 ;	The use of the Z-axis as a hypothetical axis is canceled.



4.16 VARIABLE LEAD THREADING (G34)

Specifying an increment or a decrement value for a lead per screw revolution enables variable lead threading to be performed.

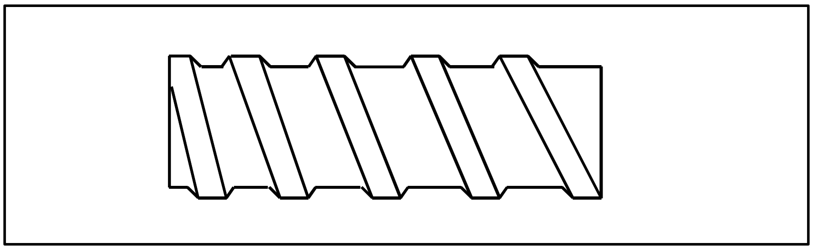


Fig. 4.16 (a) Variable lead screw

Format

G34 IP_ F_ K_ Q_ ;
- IP_ : End point
- F_ : Lead in longitudinal axis direction at the start point
- K_ : Increment and decrement of lead per spindle revolution
- Q_ : Shift amount of starting angle of thread cutting

Explanation

Address other than K are the same as in straight/taper thread cutting with G32.
The K value depends on the increment system of the reference axis, as indicated in Table 4.16 (a).
If the specified K value exceeds the range indicated in Table 4.16 (a), if the maximum lead is exceeded after a change due to the K value, or if the lead value is negative, an alarm PS0313 will be issued.

Table 4.16 (a) Range of valid K values

Increment system of reference axis	Metric input (mm/rev)	Inch input (inch/rev)
IS-A	±0.001 to ±500.000	±0.00001 to ±50.00000
IS-B	±0.0001 to ±500.0000	±0.000001 to ±50.000000
IS-C	±0.00001 to ±50.00000	±0.0000001 to ±5.0000000
IS-D	±0.000001 to ±5.000000	±0.00000001 to ±0.50000000
IS-E	±0.0000001 to ±0.5000000	±0.000000001 to ±0.050000000

> ⚠ **CAUTION**
> The "thread cutting cycle retract" is not effective for G34.

Example

Lead at the start point: 8.0 mm
Lead increment: 0.3 mm/rev

```
G34 Z-72.0 F8.0 K0.3 ;
```

4.17 CIRCULAR THREADING (G35, G36)

Using the G35 and G36 commands, a circular thread, having the specified lead in the direction of the major axis, can be machined.

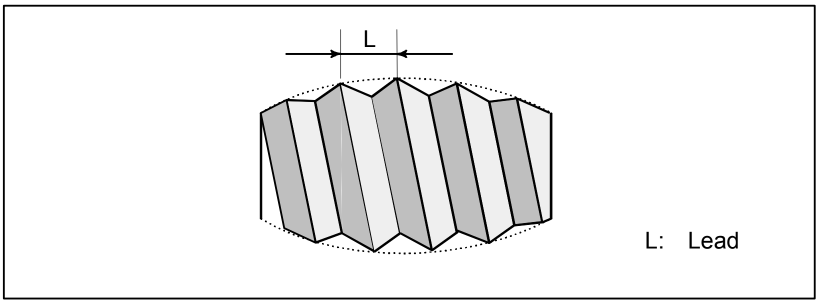


Fig. 4.17 (a) Circular threading

Format

A sample format for the G18 plane (Z-X plane) is indicated below. When using the format for the G17 plane (X-Y plane), change the addresses Z, X, K, and I to X, Y, I, and J respectively. When using the format for the G19 plane (Y-Z plane), change the addresses Z, X, K, and I to Y, Z, J, and K respectively.

M

$$\left\{\begin{matrix}\mathbf{G35}\\ \mathbf{G36}\end{matrix}\right\}\ \mathbf{X\_\ Z\_}\ \left\{\begin{matrix}\mathbf{I\_\ K\_}\\ \mathbf{R\_}\end{matrix}\right\}\ \mathbf{F\_\ Q\_\ ;}$$

G35 : Clockwise circular threading command
G36 : Counterclockwise circular threading command

X, Z : Specify the arc end point (in the same way as for G02, G03).
I, K : Specify the arc center relative to the start point, using relative coordinates (in the same way as for G02, G03).
R : Specify the arc radius.
F : Specify the lead in the direction of the major axis.
Q : Specify the shift of the threading start angle
(0° to 360°, with least input increment of 0.001)
(The value can be programmed with a decimal point.)

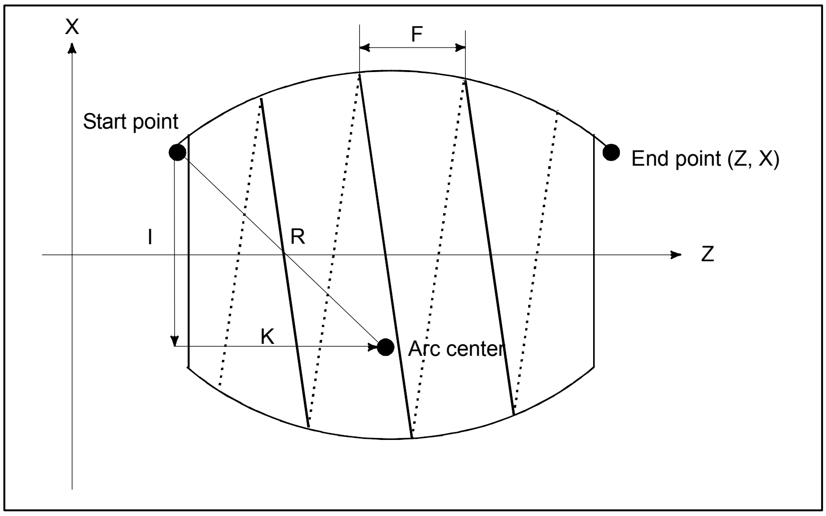
T

$$\left\{\begin{matrix}\textbf{G35}\\\textbf{G36}\end{matrix}\right\}\ \textbf{X(U)\_ Z(W)\_}\ \left\{\begin{matrix}\textbf{I \_ K \_}\\\textbf{R \_}\end{matrix}\right\}\ \textbf{F \_ Q \_ ;}$$

G35 : Clockwise circular threading command
G36 : Counterclockwise circular threading command

X(U) : Specify the arc end point (in the same way as for G02, G03).
Z(W)

I, K : Specify the arc center relative to the start point, using relative coordinates (in the same way as for G02, G03).
R : Specify the arc radius.
F : Specify the lead in the direction of the major axis.
Q : Specify the shift of the threading start angle
(0° to 360°, with least input increment of 0.001)
(The value cannot be programmed with a decimal point.)



Explanation

- Specifying the arc radius

If R is specified with I and K, only R is effective.

- Shift angle

If an angle greater than 360° is programmed, it is set to 360°.

M

- Specifying shift angle Q

To specify the shift angle Q, set bit 0 (GQS) of parameter No. 3451 to 1.

T

- Automatic tool compensation

The G36 command is used to specify the following two functions: Automatic tool compensation X and counterclockwise circular threading. The function for which G36 is to be used depends on bit 3 (G36) of parameter No. 3405.

- When parameter G36 is set to 0, the G36 command is used for automatic tool compensation X.
- When parameter G36 is set to 1, the G36 command is used for counterclockwise circular threading.

G37.1 can be used to specify automatic tool compensation X and G37.2 can be used to specify automatic tool compensation Z.

(Specification method)
G37.1 X_
G37.2 Z_

- G code when parameter G36 (bit 3 of No. 3405) is set to 1

G code	G code group	Function
G35	01	Clockwise circular threading
G36		Counterclockwise circular threading
G37	00	Automatic tool compensation Z
G37.1		Automatic tool compensation X
G37.2		Automatic tool compensation Z

Limitation

- Range of specifiable arc

An arc must be specified such that it falls within a range in which the major axis of the arc is always the Z-axis or always the X-axis, as shown in Fig. 4.17 (b), and (c). If the arc includes a point at which the major axis changes from the X-axis to Z-axis, or vice versa, as shown in Fig. 4.17 (d), an alarm PS5058 is issued.

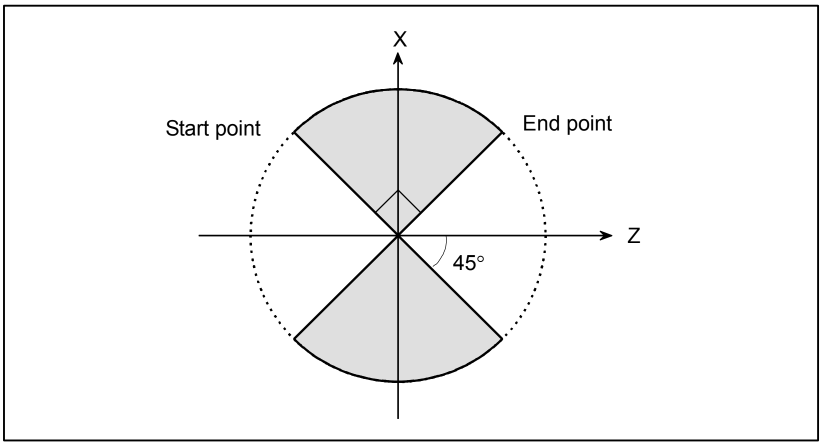


Fig. 4.17 (b) Range in which the Z-axis is the major axis

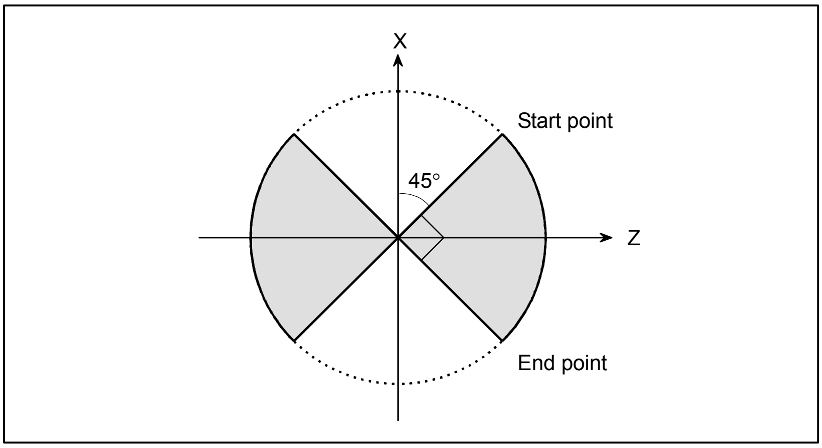


Fig. 4.17 (c) Range in which the X-axis is the major axis

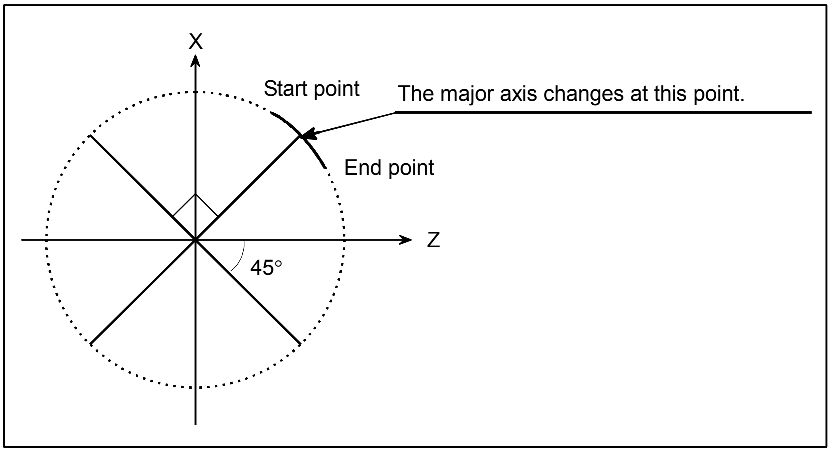


Fig. 4.17 (d) Example of arc specification which causes an alarm

- End point not on an arc

If the end point is not on an arc, a movement on an axis is made to a position of which coordinate matches the corresponding coordinate of the end point. Then, a movement is made on another axis to reach the end point.

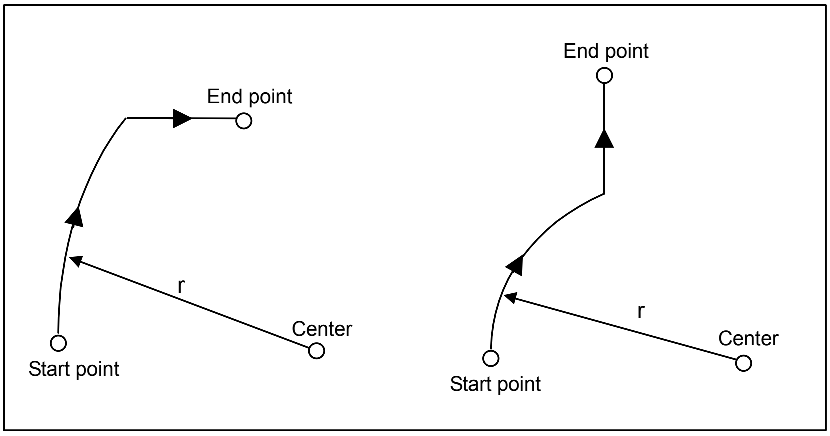


Fig. 4.17 (e) Movement when the end point is not on an arc

4.18 SKIP FUNCTION (G31)

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format

G31 IP ;

G31 : One-shot G code (If is effective only in the block in which it is specified)

Explanation

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #5061 to #5080, as follows. For systems having more than 20 axes, #100151 to #100182 are used.

#5061 X axis coordinate value
#5062 Y axis coordinate value
:
#5080 20th axis coordinate value

⚠ **CAUTION**

Disable feedrate override, dry run, and automatic acceleration/deceleration (however, these become available by setting the parameter SKF No.6200#7 to 1.) when the feedrate per minute is specified, allowing for an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per rotation is specified.

NOTE

If G31 command is issued while cutter or tool nose radius compensation is applied, an alarm PS0035 is displayed. Cancel the cutter compensation with the G40 command before the G31 command is specified.

Example

- The next block to G31 is an incremental command

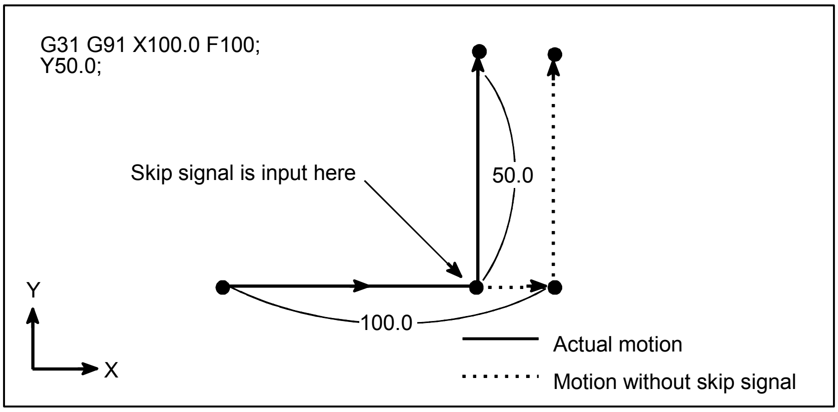


Fig. 4.18 (a) The next block is an incremental command

- The next block to G31 is an absolute command for 1 axis

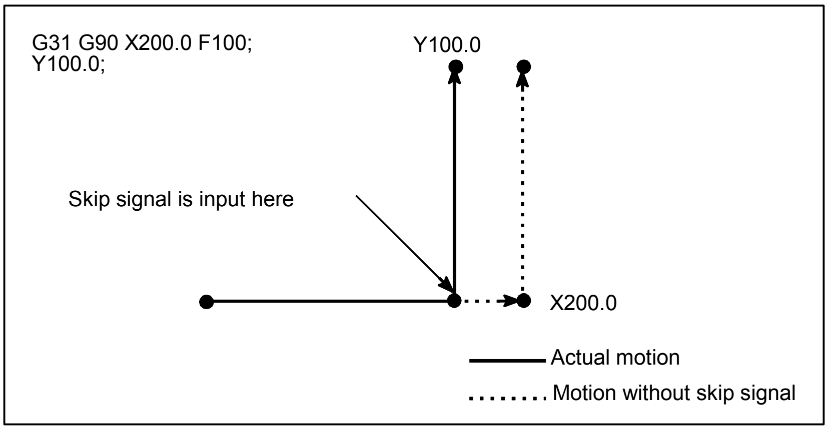


Fig. 4.18 (b) The next block is an absolute command for 1 axis

- The next block to G31 is an absolute command for 2 axes

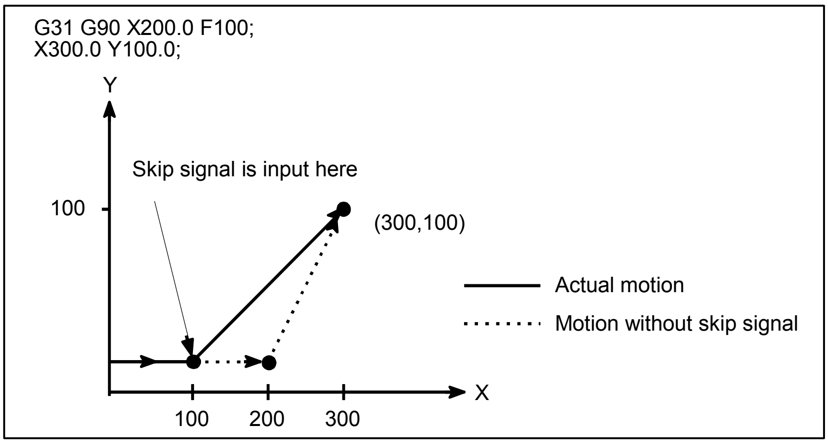


Fig. 4.18 (c) The next block is an absolute command for 2 axes

4.19 MULTI-STEP SKIP (G31)

In a block specifying P1 to P4 after G31, the multi-step skip function stores coordinates in a custom macro variable when a skip signal (4-point or 8-point ; 8-point when a high-speed skip signal is used) is turned on. In the block where Q1 to Q4 are specified after G04, dwell can be skipped when skip signals (four or eight signals, or eight signals when high-speed skip signals are used) are input.
A skip signal from equipment such as a fixed-dimension size measuring instrument can be used to skip programs being executed.
In plunge grinding, for example, a series of operations from rough machining to spark-out can be performed automatically by applying a skip signal each time rough machining, semi-fine machining, fine-machining, or spark-out operation is completed.

Format

Move command
G31 IP_ F_ P_ ;
IP_ : End point
F_ : Feedrate
P_ : P1 to P4
Dwell
G04X(U,P)_ (Q_);
X(U,P)_ : Dwell time
Q_ : Q1 to Q4

Explanation

Multi-step skip is caused by specifying P1, P2, P3, or P4 in a G31 block. For an explanation of selecting (P1, P2, P3, or P4), refer to the manual supplied by the machine tool builder.
Specifying Q1, Q2, Q3, or Q4 in G04 (dwell command) enables dwell skip in a similar way to specifying G31. A skip may occur even if Q is not specified. For an explanation of selecting (Q1, Q2, Q3, or Q4), refer to the manual supplied by the machine tool builder.

- Correspondence to skip signals

Parameter Nos. 6202 to 6205 can be used to specify whether the 4-point or 8-point skip signal is used (when a high-speed skip signal is used). Specification is not limited to one-to-one correspondence. It is possible to specify that one skip signal correspond to two or more Pn's or Qn's (n=1, 2, 3, 4). Also, parameters DS1 and DS8 (No. 6206#0 and #7) can be used to specify dwell.

> ⚠ **CAUTION**
> Dwell is not skipped when Qn is not specified and parameters DS1-DS8 (No. 6206#0-#7) are not set.

4.20 HIGH-SPEED SKIP SIGNAL (G31)

The skip function operates based on a high-speed skip signal (connected directly to the NC; not via the PMC) instead of an ordinary skip signal. In this case, up to eight signals can be input.
Delay and error of skip signal input is 0 - 2 msec at the NC side (not considering those at the PMC side).
This high-speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement.
For details, refer to the appropriate manual supplied from the machine tool builder.

Format

G31 IP ;

G31; One-shot G code (If is effective only in the block in which it is specified)

4.21 THREE-DIMENSIONAL CIRCULAR INTERPOLATION

Overview

Specifying an intermediate and end point on an arc enables circular interpolation in a 3-dimensional space.

Format

The command format is as follows:

G02.4 X_{X1} Y_{Y1} Z_{Z1} $\alpha_{\alpha 1}$ $\beta_{\beta 1}$ **;** First block (mid-point of the arc)

X_{X1} Y_{Y1} Z_{Z1} $\alpha_{\alpha 1}$ $\beta_{\beta 1}$ **;** Second block (end point of the arc)

α,β : Arbitrary axes other than the 3-dimensional circular interpolation axis (up to two axes)
To be specified if necessary.

G03.4 can also be specified instead of G02.4.
G03.4 performs the same operation as G02.4.

Explanation

- G code group

G02.4 and G03.4 are modal G codes of group 01. They therefore remain effective until another G code in group 01 is specified.

- Start point, mid-point, and end point

An arc in a 3-dimensional space is uniquely defined with its start point (current position) and a specified intermediate point and end point, as shown below. Two command blocks are used to define this arc. The first command block specifies the tool path between the start point and intermediate point. The second command block specifies the tool path between the intermediate point and end point.

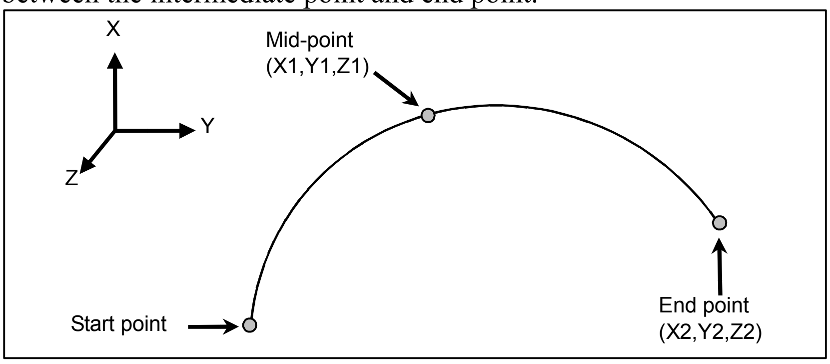


Fig. 4.21 (a) Start, Mid, and End Points

If the modal code is changed by specifying a code such as G01 with the end point not specified, the arc cannot be obtained, and alarm PS5432 is issued. During MDI operation, alarm PS5432 is also issued if a cycle start is applied with only the mid-point specified.

- Movement along axes other than the 3-dimensional circular interpolation axis

In addition to the 3-dimensional circular interpolation axis (X/Y/Z), up to two arbitrary axes (α/β) can be specified at a time. If / are omitted from the first block (mid-point specification) and are specified only in the second block (end point specification), the tool moves to the specified point along the α/β axes during movement from the mid-point of the arc to the end point. If α/β are omitted from the second block (end point specification) and are specified only in the first block (mid-point specification), the tool moves to the specified point along the α/β axes during movement from the start point of the arc to the mid-point.

- Incremental commands

With an incremental command, the position of the mid-point relative to the start point must be specified in the first block, and the position of the end point relative to the mid-point must be specified in the second block.

- Direction of rotation

The direction of rotation cannot be specified. The movement is the same regardless of whether G02.4 or G03.4 is specified.

- Single block

When operation is performed using a single block, one cycle start causes movement from the start point to the end point. A single-block stop is not performed between the first block (mid-point specification) and the second block (end point specification).

- Start point assumed if 3-dimensional circular interpolations are specified consecutively

If 3-dimensional circular interpolations are specified consecutively, the end point in one interpolation is assumed to be the start point in the next interpolation.

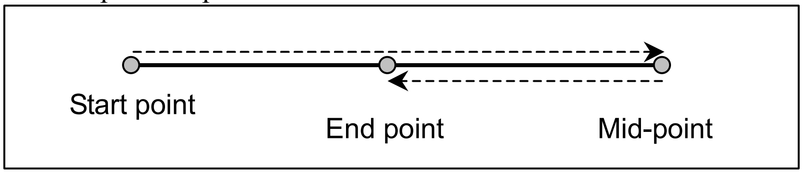
- Velocity commands

As the velocity command, specify the tangential velocity along the arc in the three-dimensional space.

Limitation

- Cases in which linear interpolation is performed

- f the start point, mid-point, and end-point are on the same line, linear interpolation is performed.
- If the start point coincides with the mid-point, the mid-point coincides with the end point, or the end point coincides with the start point, linear interpolation is performed up to the end point.
- If the start point, mid-point, and end-point are on the same line and the end point lies between the start point and the mid-point, the tool first moves with linear interpolation from the start point to the mid-point, then returns from the mid-point to the end point with linear interpolation. Thus, the tool always passes through the specified point.



- Whole circles

A whole circle (360° arc) cannot be specified. (This corresponds to the case in which linear interpolation is performed, as described earlier.)

- Compensation functions

Before using this function, cancel the compensation functions of group 07, such as cutter radius compensation.

- Manual absolute

While this function is in use, manual intervention is not possible with the manual absolute switch set to the ON position. If intervention is performed, alarm PS0713 is issued when operation restarts.

- Restrictions on commands

In the three-dimensional circular interpolation mode, the functions listed below can be used, but the states of these functions must not be modified.

- Inch input/metric input (An alarm is issued if a state modification is made with G20 or G21.)
- Mirror image (The state of a signal must not be changed.)
- One-digit F code feed (No feedrate change must be made using the manual handle.)

- Unusable commands

In the three-dimensional circular interpolation mode, the functions listed below must not be specified. Otherwise, an alarm is issued.

- Exponential interpolation..G02.3,G03.3
- Dwell G04
- High-speed machining G05
  (Excluding G05P10000 and G05P0)
- AI contour control G05.1Q1,G5.1Q0
- Hypothetical axis interpolation..G07
- Cylindrical interpolation...G07.1
- Advanced preview control..G08
  (Use AI high-precision contour control.)
- Polar coordinate interpolation.....................................G12.1,G13.1
- Polar coordinate command G15,G16
- Reference position return check G27
- Reference position return...G28
- 2nd reference position return...G30
- 3rd, 4th reference position return...G30
- Skip...G31
- Threading..G33
- Automatic tool length measurement..G37
- 3-dimensional cutter compensation G41
- Tool offset.. G45,G46,G47,G48
- Programmable mirror image.......................................G50.1,G51.1
- Local coordinate system G52
- Machine coordinate system G53
- Single direction positioning...G60
- Tapping mode G63
- Macro call (Subprogram call is possible.) G65,G66,G67
- Figure copy G72.1,G72.2
- Canned cycle..............................G73-G79,G80,G81-G89,G98,G99
- Workpiece coordinate system setting G92
- Workpiece coordinate system preset G92.1
- Feed per revolution G95
- Constant surface speed control G96,G97
- NURBS interpolation..G06.2
- Workpiece coordinate system setting G54-G59,G54.1
- 3-dimensional coordinate conversion G68
- Coordinate system rotation G68
- Scaling G50,G51
- Programmable mirror image.......................................G50.1,G51.1
- Exact stop mode..G61
- Cutter or tool nose radius compensation..... G38,G39,G40,G41,G42
- 3-dimensional cutter compensation G40,G41.2,G41.3,G42.2
- Data setting G10
- Exact stop G09
- Auxiliary function
- 2nd auxiliary function
- Spindle function
- Tool function

- Unusable functions

If the following function is specified in the three-dimensional circular interpolation mode, a warning is output:

- MDI intervention

If any of the following functions is specified in the three-dimensional circular interpolation mode, the alarm (PS5196) is issued:

- Manual operation interrupt
- Tool retract and recover

In the three-dimensional circular interpolation mode, the following functions cannot be used:

- Sequence number comparison and stop (Stop operation cannot be performed with a sequence number in the three-dimensional circular interpolation mode.)
- Index table indexing
- Rotary axis control
- Macro executor (Execution macro)
- Manual handle interruption
- Optional chamfering/corner R

- Other limitations

When the following function is used, three-dimensional circular interpolation cannot be used:

- Angular axis control

A limitation may be imposed on other NC command combinations. See the description of each function.

5 FEED FUNCTIONS

5.1 OVERVIEW

The feed functions control the feedrate of the tool. The following two feed functions are available:

- Feed functions

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse feedrate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

- Override

Override can be applied to a rapid traverse rate or cutting feedrate using the switch on the machine operator's panel.

- Automatic acceleration/deceleration

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 5.1(a)).

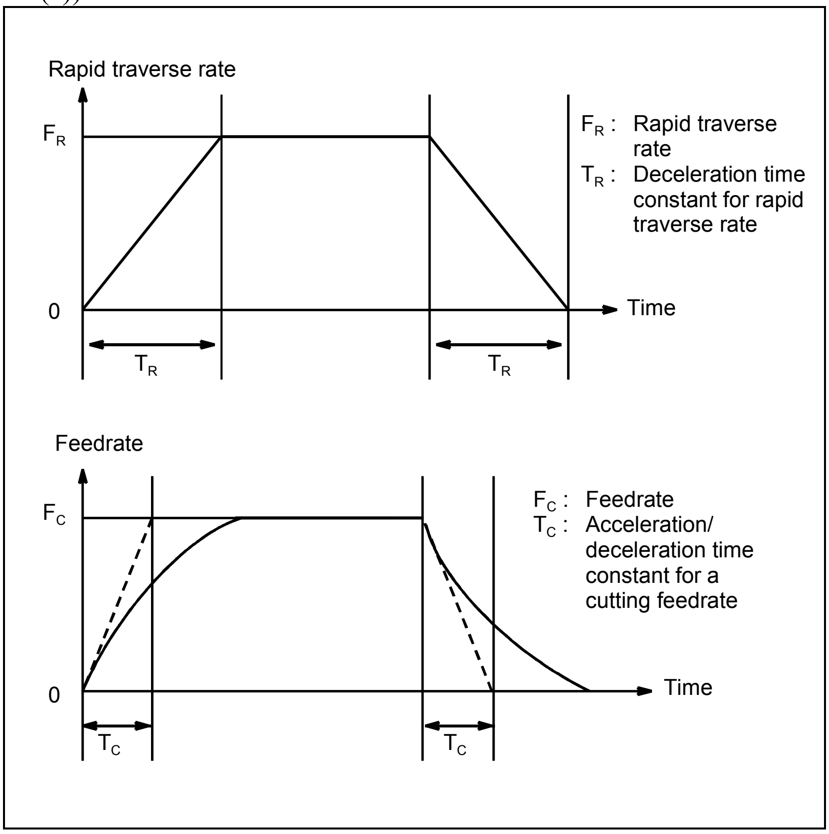


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- Tool path in a cutting feed

When the movement direction changes between a specified block and the next block during cutting feed, the tool path may be rounded because of the relationship between the time constant and feedrate (Fig. 5.1(b)).

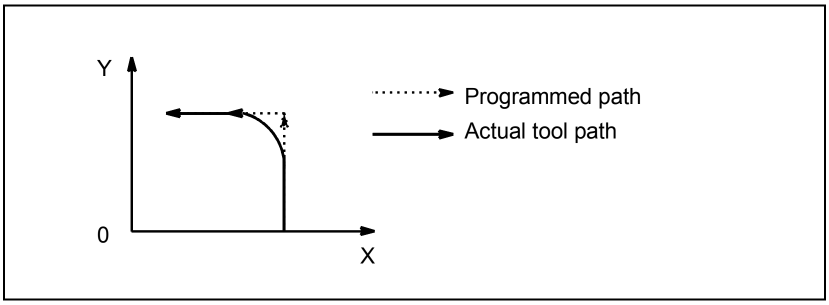


Fig. 5.1 (b) Example of tool path between two blocks

In circular interpolation, a radial error occurs (Fig. 5.1(c)).

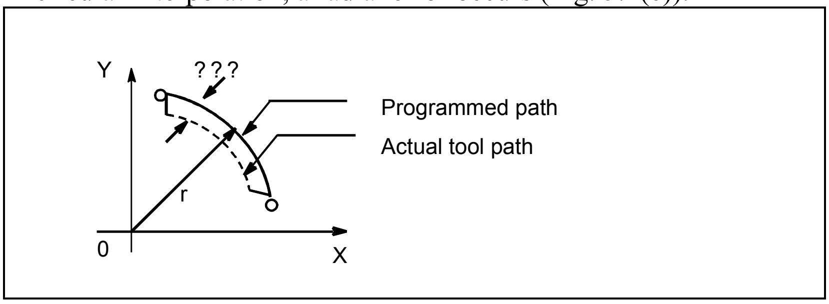


Fig. 5.1 (c) Example of radial error in circular interpolation

The rounded-corner path shown in Fig. 5.1(b) and the error shown in Fig. 5.1 (c) depend on the feedrate. So, the feedrate needs to be controlled for the tool to move as programmed.

5.2 RAPID TRAVERSE

Format

G00 IP_ ;

G00	: G code (group 01) for positioning (rapid traverse)
IP_	: Dimension word for the end point

Explanation

The positioning command (G00) positions the tool by rapid traverse. In rapid traverse, the next block is executed after the specified feedrate becomes 0 and the servo motor reaches a certain range set by the machine tool builder (in-position check).
A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse feedrate need be programmed.
The following overrides can be applied to a rapid traverse rate with the switch on the machine operator's panel:F0, 25%, 50%, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
For detailed information, refer to the appropriate manual of the machine tool builder.

5.3 CUTTING FEED

Overview

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are commanded with numbers after the F code.
In cutting feed, the next block is executed so that the feedrate change from the previous block is minimized.

M

Four modes of specification are available:

1. Feed per minute (G94)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G95)
   After F, specify the amount of feed of the tool per spindle revolution.
3. Inverse time feed (G93)
   Specify the inverse time (FRN) after F.
4. One-digit F code feed
   Specify a desired one-digit number after F. Then, the feedrate set with the CNC for that number is set.

T

Two modes of specification are available:

1. Feed per minute (G94)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G95)
   After F, specify the amount of feed of the tool per spindle revolution.

Format

M

Feed per minute
- **G94 ;** G code (group 05) for feed per minute
- **F_ ;** Feedrate command (mm/min or inch/min)

Feed per revolution
- **G95 ;** G code (group 05) for feed per revolution
- **F_ ;** Feedrate command (mm/rev or inch/rev)

Inverse time feed (G93)
- **G93 ;** Inverse time feed command G code (05 group)
- **F_ ;** Feedrate command (1/min)

One-digit F code feed
- **Fn ;**
- **n :** Number from 1 to 9

T

Feed per minute
G94 ; G code (group 05) for feed per minute
F_ ; Feedrate command (mm/min or inch/min)
Feed per revolution
G95 ; G code (group 05) for feed per revolution
F_ ; Feedrate command (mm/rev or inch/rev)

Explanation

- Direction of the cutting feedrate

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

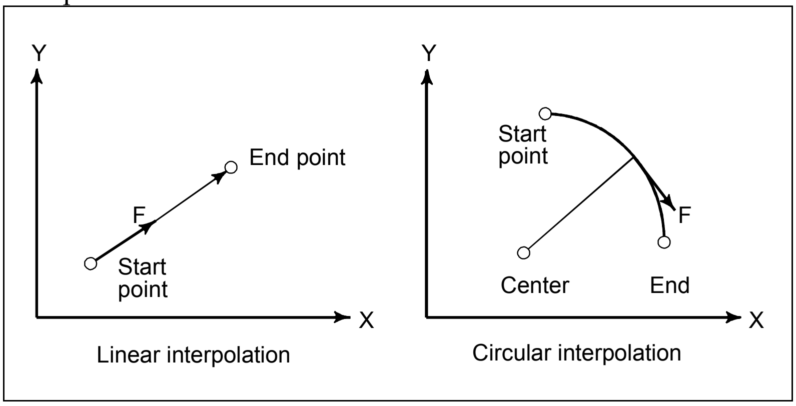


Fig. 5.3 (a) Tangential feedrate (F)

- Feed per minute

After specifying G code for feed per minute (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. G code for feed per minute is a modal code. Once a G code for feed per minute is specified, it is valid until G code for feed per revolution (feed per revolution) is specified.

M

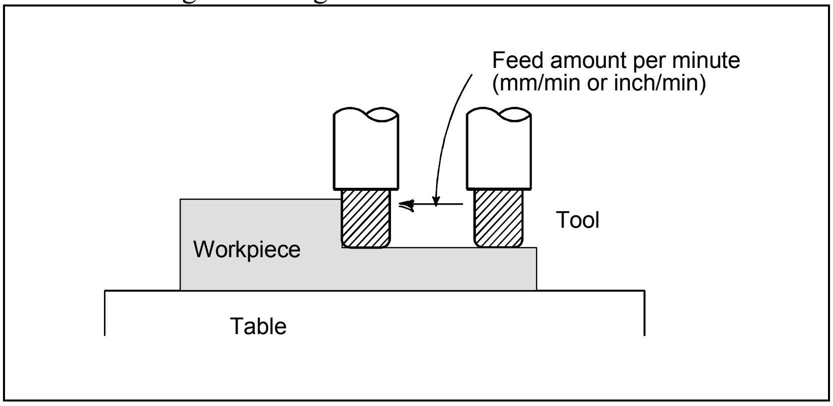
At power-on, the feed per minute mode is set.

T

At power-on, the feed per revolution mode is set.

An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

- For milling machining



- For lathe cutting

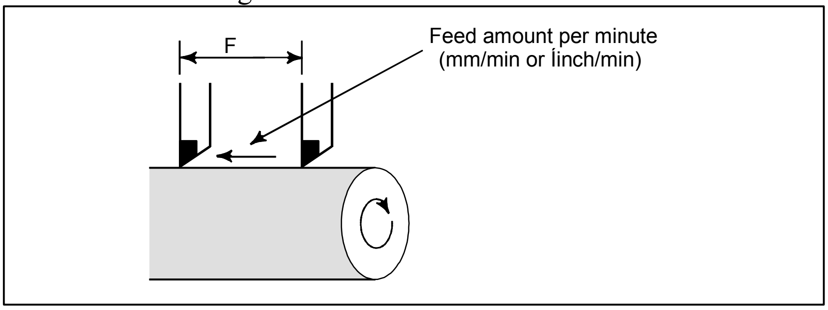


Fig. 5.3 (b) Feed per minute

> ⚠ **CAUTION**
> No override can be used for some commands such as for threading.

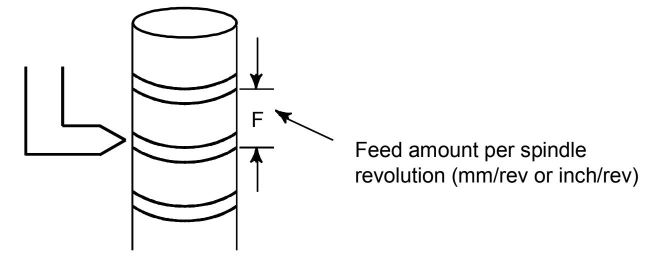
- Feed per revolution

After specifying G code for feed per revolution (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G code for feed per revolution is a modal code. Once a G code for feed per revolution is specified, it is valid until G code for feed per minute (feed per minute) is specified.
An override from 0% to 254% (in 1% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

T

If parameter NPC (No. 1402#0) has been set to 1, feed per revolution commands can be specified even when a position coder is not being used. (The CNC converts feed per revolution commands to feed per minute commands.)

- For milling machining



- For lathe cutting

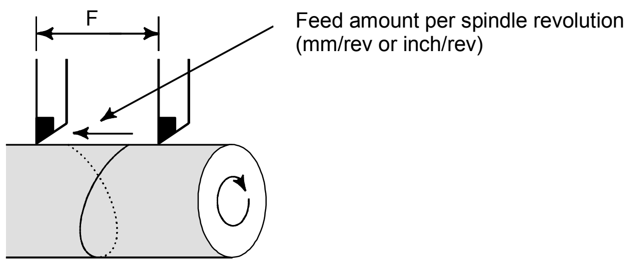


Fig. 5.3 (c) Feed per revolution

> ⚠ **CAUTION**
> When the speed of the spindle is low, feedrate fluctuation may occur.
> The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

- Inverse time feed

M

When G code for inverse time feed is specified, the inverse time specification mode (G93 mode) is set. Specify the inverse time (FRN) with an F code.
A value from 0.001 to 9999.999 can be specified as FRN, regardless of whether the input mode is inches or metric, or the increment system is IS-B or IS-C.

F code specification value	FRN
F1	0.001
F1 (*1)	1.000
F1.0	1.000
F9999999	9999.999
F9999 (*1)	9999.000
F9999.999	9999.999

> **NOTE**
> *1 Value specified in fixed-point format with parameter DPI (No. 3401#0) set to 1

G code for inverse time feed is a modal G code and belongs to group 05 (includes G code for feed per revolution and G code for feed per minute).
When an F value is specified in inverse time specification mode and the feedrate exceeds the maximum cutting feedrate, the feedrate is clamped to the maximum cutting feedrate.
In the case of circular interpolation, the feedrate is calculated not from the actual amount of movement in the block but from the arc radius. This means that actual machining time is longer when the arc radius is longer than the arc distance and shorter when the arc radius is shorter than the arc distance. Inverse time feed can also be used for cutting feed in a canned cycle.

NOTE

1. In the inverse time specification mode, an F code is not handled as a modal code and therefore needs to be specified in each block. If an F code is not specified, alarm (PS0011 (indicating that cutting feedrate specification is missing)) is issued.
2. When F0 is specified in inverse time specification mode, alarm (PS0011 (indicating that cutting feedrate specification is missing)) is issued.
3. Inverse time feed cannot be used when PMC axis control is in effect.
4. If the calculated cutting feedrate is smaller than the allowable range, alarm (PS0011 (indicating that cutting feedrate specification is missing)) is issued.

- One-digit F code feed

M

When a one-digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameter (Nos. 1451 to 1459) is used. When F0 is specified, the rapid traverse rate is applied.
The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing One-digit F feedrate on the machine operator's panel, then by rotating the manual pulse generator.
The increment/decrement, ΔF, in feedrate per scale of the manual pulse generator is as follows:

$$\Delta F = \frac{F\,\max}{100X}$$

Fmax : Feedrate upper limit for F1-F4 set by parameter (No.1460), or feedrate upper limit for F5-F9 set by parameter (No.1461)

X : Any value of 1-127 set by parameter (No.1450)

The feedrate set or altered is kept even while the power is off. The current feed rate is displayed on the LCD screen.

- Cutting feedrate clamp

Parameter No. 1430 can be used to specify the maximum cutting feedrate for each axis. When the cutting feedrate along an axis exceeds the maximum feedrate for the axis as a result of interpolation, the cutting feedrate is clamped to the maximum feedrate.

Example

- **For linear interpolation (G01)**

$$FRN = \frac{1}{time(min)} = \frac{feedrate}{distance}$$

Feedrate:	mm/min	(for metric input)
	inch/min	(for inch input)
Distance:	mm	(for metric input)
	inch	(for inch input)

- To end a block in 1 (min)

$$FRN = \frac{1}{time(\text{min})} = \frac{1}{1(\text{min})} = 1$$

Specify F1.0.

- To end a block in 10 (sec)

$$FRN = \frac{1}{time(\text{sec})/60} = \frac{1}{10/60(\text{sec})} = 6$$

Specify F6.0.

- To find the movement time required when F0.5 is specified

$$TIME(\text{min}) = \frac{1}{FRN} = \frac{1}{0.5} = 2$$

2 (min) is required.

- To find the movement time required when F10.0 is specified

$$TIME(\text{min}) = \frac{1 \times 60}{FRN} = \frac{60}{10} = 6$$

6 (sec) is required.

- **For circular interpolation (G02, G03)**

$$FRN = \frac{1}{time(\text{min})} = \frac{feedrate}{arcradius}$$

Feedrate:	mm/min	(for metric input)
	inch/min	(for inch input)
Arc radius:	mm	(for metric input)
	inch	(for inch input)

NOTE

In the case of circular interpolation, the feedrate is calculated notfrom
the actual amount of movement in the block but from the arcadias.

Reference

See Appendix D for range of feedrate command value.

5.4 CUTTING FEEDRATE CONTROL

Cutting feedrate can be controlled, as indicated in Table 5.4 (a).

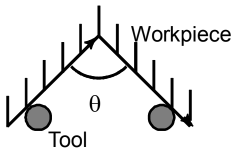
Table 5.4 (a) Cutting Feedrate Control

Function name		G code	Validity of G code	Description
Exact stop		G09	This function is valid for specified blocks only.	The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.
Exact stop mode		G61	Once specified, this function is valid until G62, G63, or G64 is specified.	The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.
Cutting mode		G64	Once specified, this function is valid until G61, G62, or G63 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed.
Tapping mode		G63	Once specified, this function is valid until G61, G62, or G64 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are invalid.
Automatic corner override	Automatic override for inner corners	G62	Once specified, this function is valid until G61, G63, or G64 is specified.	When the tool moves along an inner corner during cutter compensation, override is applied to the cutting feedrate to suppress the amount of cutting per unit of time so that a good surface finish can be produced.
	Internal circular cutting feedrate change	-	This function is valid in the cutter compensation mode, regardless of the G code.	The internal circular cutting feedrate is changed.

NOTE

1 The purpose of in-position check is to check that the servo motor has reached within a specified range (specified with a parameter by the machine tool builder).
In-position check is not performed when parameter NCI (No. 1601#5) is set to 1.

2 Inner corner angle θ: $2° < \theta \leq \alpha \leq 178°$
(α is a set value)



Format

Exact stop	**G09 IP_;**
Exact stop mode	**G61;**
Cutting mode	**G64;**
Tapping mode	**G63;**
Automatic corner override	**G62;**

5.4.1 Exact Stop (G09, G61), Cutting Mode (G64), Tapping Mode (G63)

Explanation

The inter-block paths followed by the tool in the exact stop mode, cutting mode, and tapping mode are different (Fig. 5.4.1 (a)).

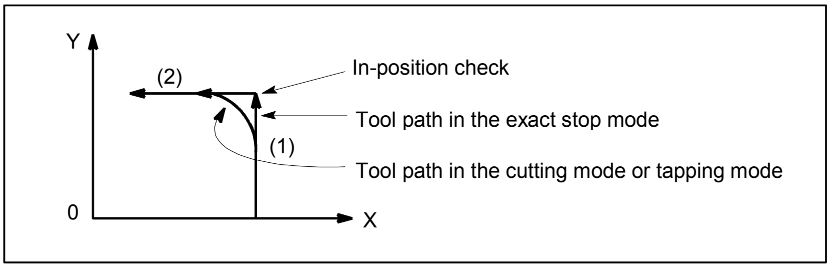


Fig. 5.4.1 (a) Example of tool paths from block (1) to block (2)

> ⚠ **CAUTION**
> The cutting mode (G64 mode) is set at power-on or system clear.

5.4.2 Automatic Corner Override

When cutter compensation is performed, the movement of the tool is automatically decelerated at an inner corner and internal circular area. This reduces the load on the tool and produces a smoothly machined surface.

5.4.2.1 Automatic override for inner corners (G62)

Explanation

- Override condition

When G62 is specified, and the tool path with cutter compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.
There are four types of inner corners (Fig. 5.4.2(a)).
2°≤θ≤θp≤178° in Fig. 5.4.2(a)qp is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001,or less.

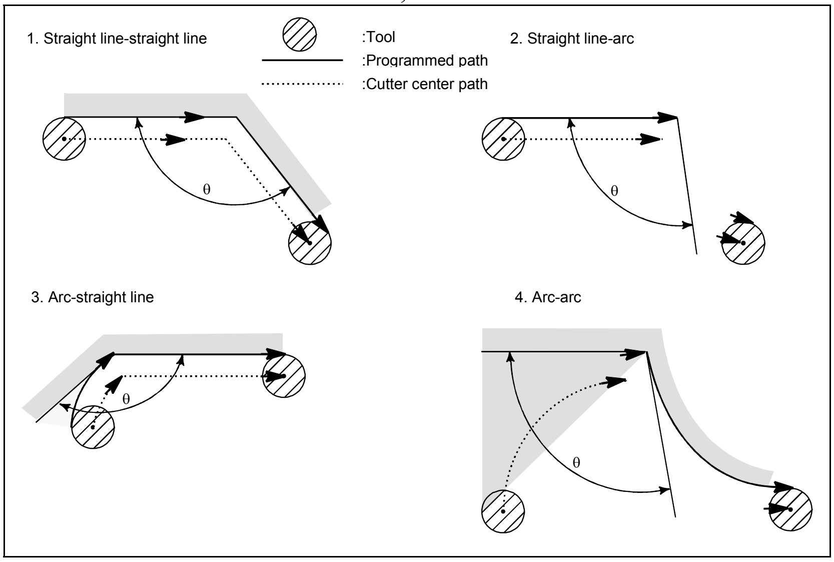


Fig. 5.4.2(a) Inner corner

- Override range

When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The distances Ls and Le, where the feedrate is overridden, are distances from points on the cutter center path to the corner (Fig. 5.4.2(b), Fig. 5.4.2(c), Fig. 5.4.2(d)). Ls and Le are set with parameter Nos. 1713 and 1714.

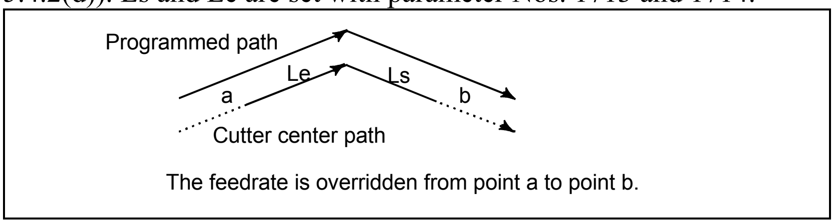


FIg. 5.4.2.1 (b) Override Range (Straight Line to Straight Line)

When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants (Fig. 5.4.2(c)).

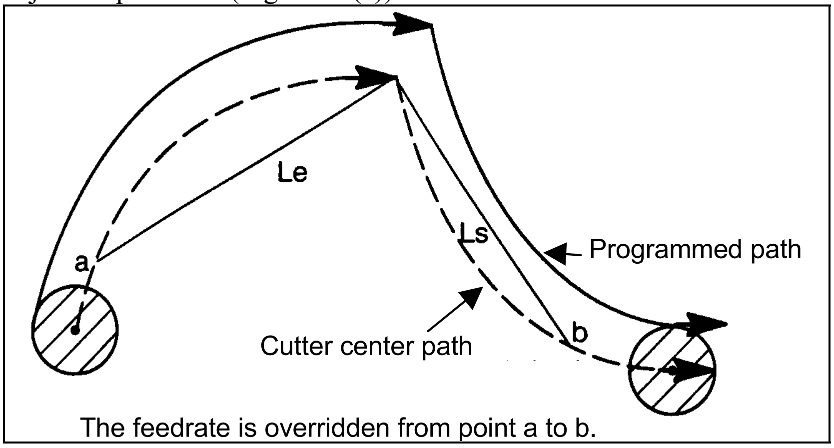


Fig. 5.4.2(c) Override Range (Arc to Arc)

Regarding program (2) of an arc, the feedrate is overridden from point a to point b and from point c to point d (Fig. 5.4.2(d)).

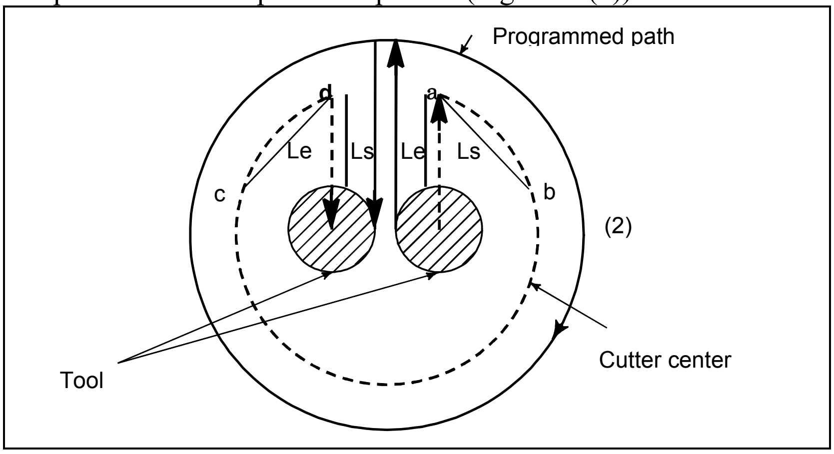


Fig. 5.4.2(d) Override Range (Straight Line to Arc, Arc to Straight Line)

- Override value

An override value is set with parameter No. 1712. An override value is valid even for dry run and one-digit F code feed specification.
In the feed per minute mode, the actual feedrate is as follows:

$$F = (\text{automatic override for inner corners}) \times (\text{feedrate override})$$

Limitation

- Acceleration/deceleration before interpolation

Override for inner corners is disabled during acceleration/deceleration before interpolation.

- Start-up/G41, G42

Override for inner corners is disabled if the corner is preceded by a start-up block or followed by a block including G41 or G42.

- Offset

Override for inner corners is not performed if the offset is zero.

5.4.2.2 Internal circular cutting feedrate change

For internally offset circular cutting, the feedrate on a programmed path is set to a specified feedrate (F) by specifying the circular cutting feedrate with respect to F, as indicated below (Fig. 5.4.2(e)). This function is valid in the cutter compensation mode, regardless of the G62 code.

$$F = \frac{Rc}{Rp}$$

Rc : Cutter center path radius
Rp : Programmed radius
It is also valid for the dry run and the one-digit F code feed command.

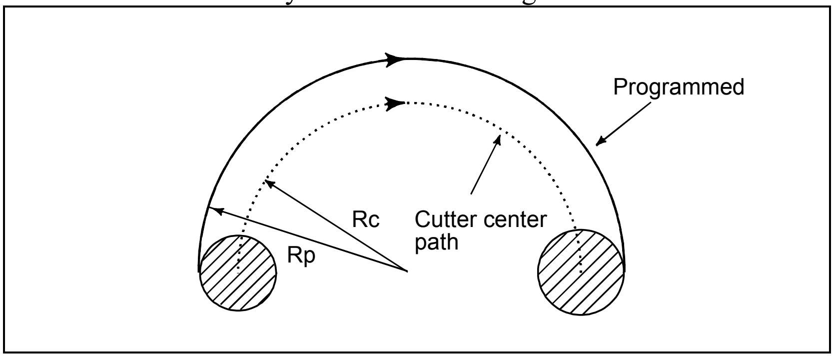


Fig. 5.4.2(e) Internal circular cutting feedrate change

If Rc is much smaller than Rp, Rc/Rp≒0; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When Rc/Rp≤MDR, the feedrate of the tool is (F×MDR).

> ⚠ **CAUTION**
> When internal circular cutting must be performed together with override for inner corners, the feedrate of the tool is as follows:
> $$F \times \frac{Rc}{Rp} \times (\text{override for the inner corners}) \times (\text{feedrate override})$$

5.5 DWELL

Format

M

G04 X_; or G04 P_;
X_ : Specify a time or spindle speed (decimal point permitted)
P_ : Specify a time or spindle speed (decimal point not permitted)

T

G04 X_ ; or G04 U_ ; or G04 P_ ;
X_ : Specify a time or spindle speed (decimal point permitted)
U_ : Specify a time or spindle speed (decimal point permitted)
P_ : Specify a time or spindle speed (decimal point not permitted)

Explanation

By specifying a dwell, the execution of the next block is delayed by the specified time. (Dwell per second)
By setting bit 1 (DWL) of parameter No. 3405 in the feed per revolution mode, the execution of the next block is delayed until the rotation count of the spindle reaches the specified number. (Dwell per revolution)

Table 5.5 (a) Command value range of the dwell time (Command by X or U)

Increment system	Command value range	Dwell time unit
IS-A	0.01 to 999999.99	s or rev
IS-B	0.001 to 99999.999	
IS-C	0.0001 to 9999.9999	
IS-D	0.00001 to 999.99999	
IS-E	0.000001 to 99.999999	

Table 5.5 (b) Command value range of the dwell time (Command by P)

Increment system	Command value range	Dwell time unit
IS-A	1 to 99999999	0.01 s or rev
IS-B	1 to 99999999	0.001 s or rev
IS-C	1 to 99999999	0.0001 s or rev
IS-D	1 to 99999999	0.00001 s or rev
IS-E	1 to 99999999	0.000001 s or rev

In the case of dwell per second, the specification unit for dwell time specified with P can be fixed at 0.001 second by setting bit 7 (DWT) of parameter No. 1015 to 1.

NOTE

1. When X, U, or P is specified without a decimal point, the specification unit does not depend on inch/metric input. Depending on whether the X-axis is present, the following increment system is used:
   - When the X-axis is present
     The increment system of the X-axis is used.
   - When the X-axis is not present
     The increment system of the reference axis is used.
2. When P is specified, bit 7 (IPR) of parameter No. 1004 exercises no influence.

M

Specify dwell also to make an exact check in the cutting mode (G64 mode).
If the specification of P and X is omitted, an exact stop occurs.

6 REFERENCE POSITION

A CNC machine tool has a special position where, generally, the tool is exchanged or the coordinate system is set, as described later. This position is referred to as a reference position.

6.1 REFERENCE POSITION RETURN

Overview

- Reference position

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function. For example, the reference position is used as a position at which tools are automatically changed. Up to four reference positions can be specified by setting coordinates in the machine coordinate system in parameters (No. 1240 to 1243).

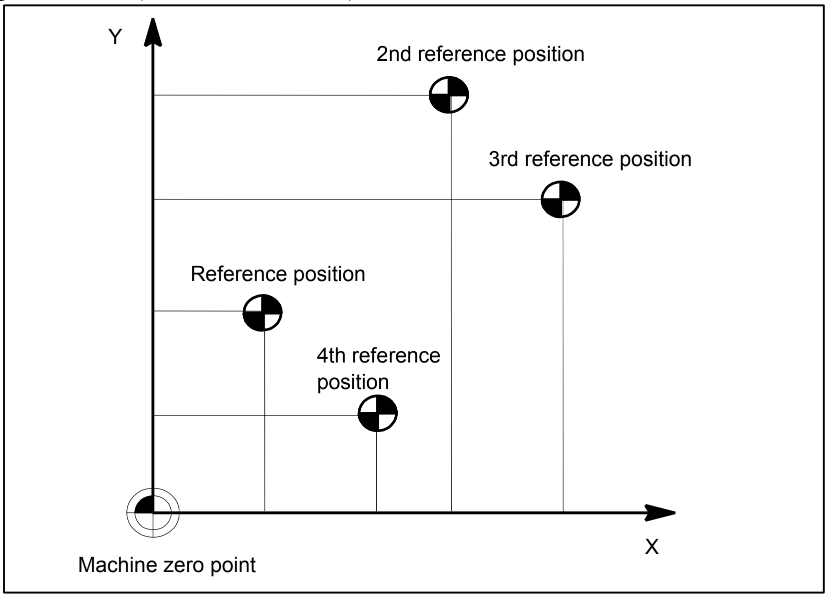


Fig. 6.1 (a) Machine zero point and reference positions

- Automatic reference position return (G28) and movement from the reference position (G29)

The automatic reference position return (G28) function returns tools automatically to the reference position via an intermediate position along a specified axis. When reference position return is completed, the lamp for indicating the completion of reference position return goes on. The return from reference position (G29) function moves tools from the reference position to a specified position via an intermediate position along a specified axis.

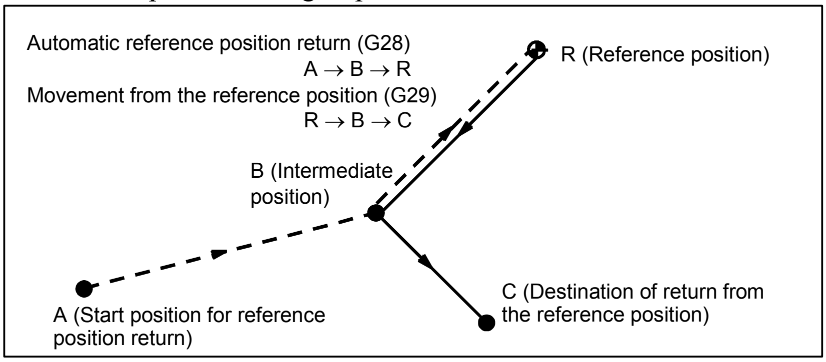


Fig. 6.1 (b) Reference position return and return form the reference position

- Reference position return check (G27)

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis for indicating the completion of reference position return goes on.
If the tool has not reached the reference position, an alarm (PS0092) "ZERO RETURN CHECK (G27) ERROR" is issued.
When no movement was made along the axis, whether the current position is the reference position is checked.

Format

- Automatic reference position return and 2nd/3rd/4th reference position return

G28 IP_;	Reference position return
G30 P2 IP_;	2nd reference position return (P2 can be omitted.)
G30 P3 IP_;	3rd reference position return
G30 P4 IP_;	4th reference position return

IP : Specify the intermediate position in the absolute coordinate system.
(absolute/incremental command)
There is no need to calculate an actual travel distance between the intermediate position and the reference position.

- Movement from reference position

G29 IP_;
IP : Specify the destination of return from the reference position in the absolute coordinate system.
(absolute/incremental command)
The intermediate position is determined by G28, G30, or G30.1 specified immediately before this command.

- Reference position return check

G27 IP_;
IP : Specify positioning to the reference position in the absolute coordinate system so as to return to the reference position. (absolute/incremental command)

Explanation

- Automatic reference position return (G28)

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis.
Therefore, for safety, the compensation functions, such as the cutter compensation and tool length compensation, should be cancelled before executing this command.
The coordinates for the intermediate position are stored in the CNC for the axes for which a value is specified in a G28 block. For the other axes, the previously specified coordinates are used.

(Example)

N1 G28 X40.0 ;
(The tool moves to the reference position along the X-axis and the intermediate position (X40.0) is stored.)
N2 G28 Y60.0 ;
(The tool moves to the reference position along the Y-axis and the intermediate position (Y60.0) is stored.)
N3 G29 X10.0 Y20.0 ;
(The tool moves to the position specified with G29 via the intermediate position (X40.0 Y60.0) previously specified with G28 along the X-axis and Y-axis.)

- 2nd, 3rd, and 4th reference position return (G30)

The 2nd, 3rd, and 4th reference position return (G30) function can be used after the reference positions are established.
The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.

- Movement from the reference position (G29)

This function is executed after the tool is returned to the reference position by G28 or G30.
For incremental programming, the command value specifies the incremental value from the intermediate point.
The tool moves to the intermediate and specified positions at the feedrate specified with a parameter.
When the workpiece coordinate system is changed after the tool reaches the reference position through the intermediate point by the G28 command, the intermediate point also shifts to a new coordinate system. If G29 is then commanded, the tool moves to the commanded position through the intermediate point which has been shifted to the new coordinate system.
The same operations are performed also for G30 and G30.1 commands.
After the power is turned on, an alarm (PS0305) is issued if an attempt is made to execute G29 (movement from the reference position) before G28 (automatic reference position return), G30 (2nd, 3rd, and 4th reference position return), or G30.1 (floating reference position return) is executed.

- Reference position return check (G27)

G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the lamp for indicating the completion of reference position return lights up.

When the tool returns to the reference position along only one axis, the lamp for the axis for indicating the completion of reference position return lights up.

After positioning, if the tool has not reached the reference position along a specified axis, an alarm (PS0092) "ZERO RETURN CHECK (G27) ERROR" is issued.

When no movement was made along the axis, whether the current position is the reference position is checked.

- Setting of the reference position return feedrate

Before a coordinate system is established with the first reference position return after power-on, the manual and automatic reference position return feedrates and automatic rapid traverse rate conform to the setting of parameter No. 1428 for each axis.

After a reference position is established upon the completion of reference position return, the manual reference position return feedrate conforms to the setting of the parameter No. 1428 for each axis.

NOTE

1 To this feedrate, a rapid traverse override (F0,25,50,100%) is applied, for which the setting is 100%.
2 After a reference position has been established upon the completion of reference position return, the automatic reference position return feedrate will conform to the ordinary rapid traverse rate.
3 When a value is set for parameter No. 1428, the feedrates conform to the parameter settings shown below.

	Before a coordinate system is established	**After a coordinate system is established**
Automatic reference position return (G28)	No. 1428	No.1420
Automatic rapid traverse (G00)	No.1428	No.1420
Manual reference position return (*1)	No.1428	No.1428 (*3)
Manual rapid traverse rate	No.1423 (*2)	No.1424

1420: Rapid traverse rate
1423: Jog feedrate
1424: Manual rapid traverse rate
1428: Reference position return feedrate

When parameter No. 1428 is set to 0, the feedrates conform to the parameter settings shown below.

	Before a coordinate system is established	**After a coordinate system is established**
Automatic reference position return (G28)	No. 1420	No.1420
Automatic rapid traverse (G00)	No.1420	No.1420
Manual reference position return (*1)	No.1424	No.1424 (*3)
Manual rapid traverse rate	No.1423 (*2)	No.1424

*1 By using JZR (bit 2 of parameter No. 1401), the manual reference position return feedrate can always be set as a jog feedrate.
*2 When RPD (bit 0 of parameter No. 1401) is 1, the setting of parameter No. 1424 (manual rapid traverse rate) is used.
When the setting of parameter No. 1424 (manual rapid traverse rate) is 0, parameter No. 1420 (rapid traverse rate) is used.
*3 When reference position return without dogs is performed in rapid traverse mode, or when manual reference position return is performed in rapid traverse mode regardless of deceleration dogs after a reference position is established, the reference position return feedrate for each of these functions (setting of DLF (bit 1 of parameter No. 1404)) is used.

Limitation

- Status the machine lock being turned on

The lamp for indicating the completion of reference position return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a reference position return check command is specified.

- When automatic reference position return (G28) is executed if no reference position is established

When automatic reference position return (G28) is executed if no reference position is established, movement from the intermediate position in a reference position direction is the same as that in manual reference position return.
(This movement is referred to as a low-speed type of automatic reference position return (G28).)
In this case, the tool moves in the direction for reference position return specified in parameter ZMIx (bit 5 of No. 1006). Therefore the specified intermediate position must be a position to which reference position return is possible.

> **NOTE**
> When automatic reference position return (G28) is executed after a reference position is established, positioning is performed from the intermediate position to the reference position. This movement is referred to as a high-speed type of automatic reference position return (G28).

- Reference position return check in an offset mode

In an offset mode, the position to be reached by the reference position return check is the position obtained by adding the offset value.
Therefore, if the position with the offset value added is not the reference position, the lamp for indicating the completion of reference position return does not light up, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.

- Lighting the lamp when the programmed position does not coincide with the reference position

When the machine tool system is an inch system with metric input, the lamp for indicating the completion of reference position return may also light up even if the programmed position is shifted from the reference position by the least setting increment. This is because the least setting increment of the machine tool system is smaller than its least command increment.

Example

G28G90X1000.0Y500.0 ;
(Programs movement from A to B. The tool moves to reference position R via intermediate position B.)
T1111 ;
(Changing the tool at the reference position)
G29X1300.0Y200.0 ;
(Programs movement from B to C. The tool moves from reference position R to C specified with G29 via intermediate position B.)

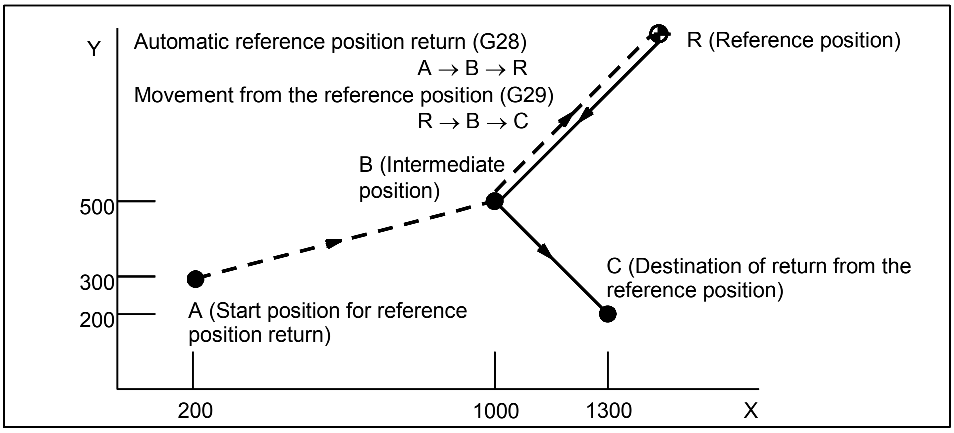


Fig. 6.1 (c) Reference position return and movement from the reference position

6.2 FLOATING REFERENCE POSITION RETURN (G30.1)

Tools ca be returned to the floating reference position.
A floating reference point is a position on a machine tool, and serves as a reference point for machine tool operation.
A floating reference point need not always be fixed, but can be moved as required.

Format

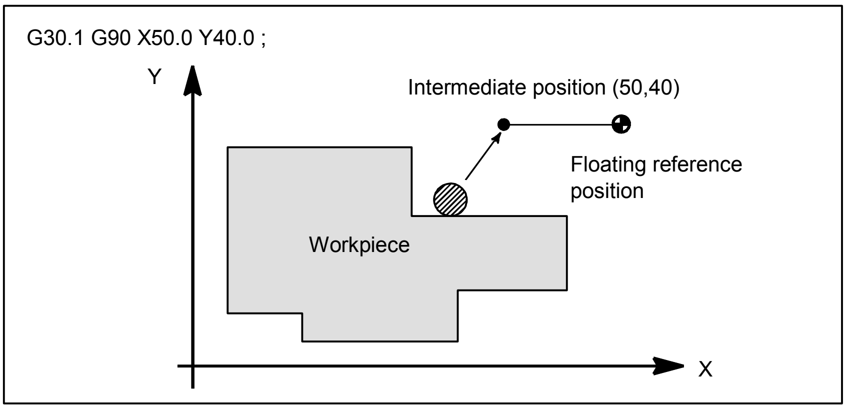
G30.1 IP_;

IP_ : Specify the intermediate position to the floating reference position in the absolute coordinate system. (absolute/incremental command)

Explanation

Generally speaking, on a machining center or milling machine, cutting tools can be replaced only at specific positions. A position where tools can be replaced is defined as the second or third reference point. Using G30 can easily move the cutting tools back to these points.
On some machine tools, the cutting tools can be replaced at any position unless they interfere with the workpiece. With these machines, the cutting tools should be replaced at a position as close to the workpiece as possible so as to minimize the machine cycle time. For this purpose, the tool change position is to be changed, depending on the figure of the workpiece. This operation can easily be performed using this function.
That is, a tool change position suitable for the workpiece is memorized as a floating reference point. Then command G30.1 can easily cause return to the tool change position.
A floating reference point becomes a machine coordinate position memorized by pressing the soft key [SET FRP] on the current positions display screen. The coordinates of the floating reference position are stored with parameter No. 1244.
The G30.1 block first positions the tool at the intermediate point along the specified axes at rapid traverse rate, then further moves the tool from the intermediate point to the floating reference point at rapid traverse rate. Before using G30.1, cancel the compensation functions, such as cutter compensation and tool length compensation.
A floating reference point is not lost even if power is turned off.
The function for movement from the reference position (G29) can be specified for moving the tool from the floating reference position.

Example


G30.1 G90 X50.0 Y40.0 ;
Y
Intermediate position (50,40)
Floating reference position
Workpiece
X

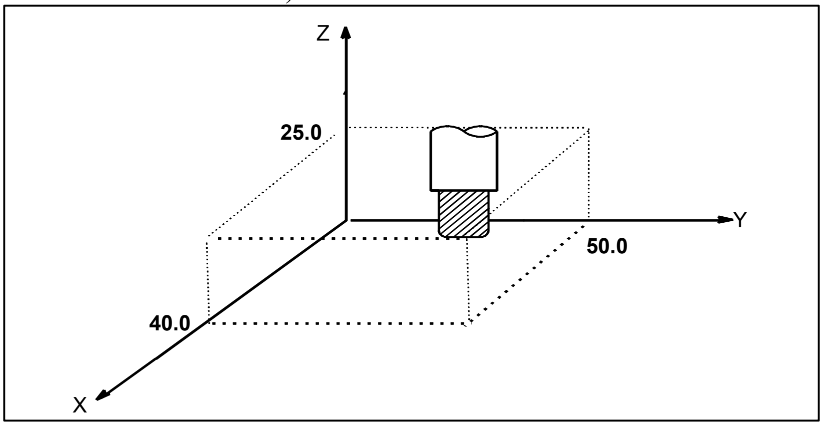
7 COORDINATE SYSTEM

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes.
When three program axes, the X-axis, Y-axis, and Z-axis, are used, coordinates are specified as follows:

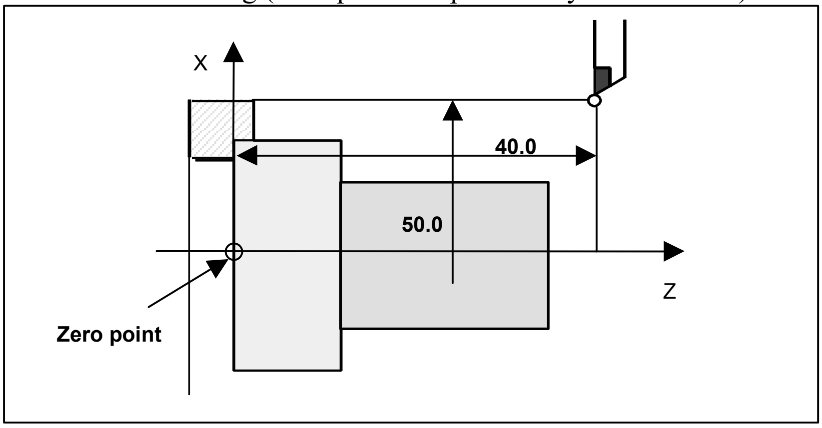
X_Y_Z_

This command is referred to as a dimension word.

- For milling machining (Tool position specified by X40.0Y50.0Z25.0)



- For lathe cutting (Tool position specified by X50.0 Z40.0)



Coordinates are specified in one of following three coordinate systems:

(1) Machine coordinate system
(2) Workpiece coordinate system
(3) Local coordinate system

The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

7.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.
A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.
A machine coordinate system is set by performing manual reference position return after power-on (see III-3.1). A machine coordinate system, once set, remains unchanged until the power is turned off.
The reference position is not always the origin of the machine coordinate system.
(See "Setting a machine coordinate system" described later.)

Format

M

(G90)G53 IP_;
IP_: Absolute dimension word

T

G53 IP_;
IP_: Absolute dimension word

Explanation

- Selecting a machine coordinate system (G53)

When a command is specified the position on a machine coordinate system, the tool moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a one-shot G code; that is, it is valid only in the block in which it is specified on a machine coordinate system. Specify an absolute command for G53. When an incremental command is specified, the G53 command is ignored. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

Limitation

- Cancel of the compensation function

When the G53 command is specified, cancel the compensation functions such as the cutter compensation, tool length compensation, tool nose radius compensation, and tool offset.

- G53 specification immediately after power-on

Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute-position detector is attached.

- Specification in the same block

M

Commands G50/G51, G50.1/G51.1, and G68/G69 cannot be specified in the same block where the G53 command is specified.

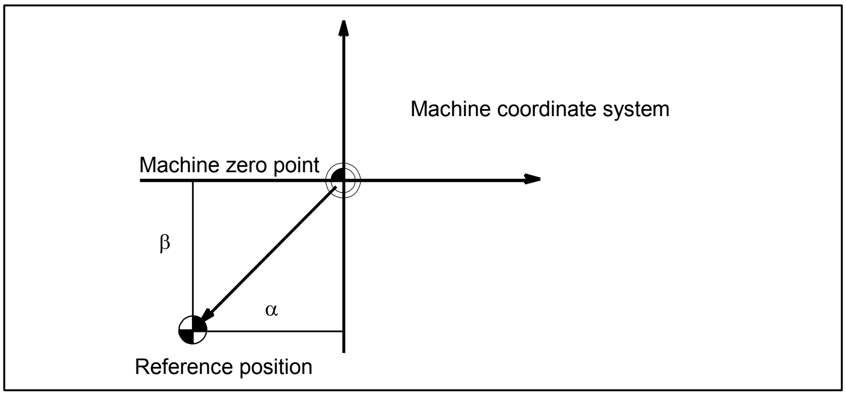
T

Commands G50/G51 (except for G code system A), G50.1/G51.1, and G68.1/G69.1 cannot be specified in the same block where the G53 command is specified.

Reference

- Setting a machine coordinate system

When manual reference position return is performed after power-on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.



7.2 WORKPIECE COORDINATE SYSTEM

Overview

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (setting a workpiece coordinate system).
A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system).
A set workpiece coordinate system can be changed by shifting its origin (changing a workpiece coordinate system).

7.2.1 Setting a Workpiece Coordinate System

A workpiece coordinate system can be set using one of three methods:

(1) Method using a workpiece coordinate system setting G code
A workpiece coordinate system is set by specifying a value in the program after a workpiece coordinate system setting G code.

(2) Automatic setting
If bit 0 of parameter ZPR No. 1201 is set to 1, a workpiece coordinate system is automatically set when manual reference position return is performed (see III-3.1.).
This function is, however, disabled when the workpiece coordinate system option is being used.

(3) Method using a workpiece coordinate system selection G code
Six workpiece coordinate systems can be set beforehand using the MDI panel. Program commands G54 to G59 can be used to select the workpiece axis to be used. (see III-12.3.4.)

When using an absolute command, establish the workpiece coordinate system in any of the above ways.

Format

- Setting a workpiece coordinate system

M

(G90) G92 IP_

T

G50 IP_

Explanation

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates.

M

If a coordinate system is set using G92 during tool length offset, a coordinate system in which the position before offset matches the position specified in G92 is set. Cutter compensation is cancelled temporarily with G92.

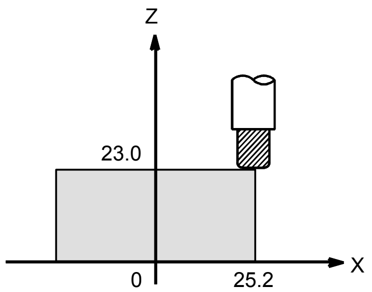
T

If IP_ is an incremental command value, the workpiece coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

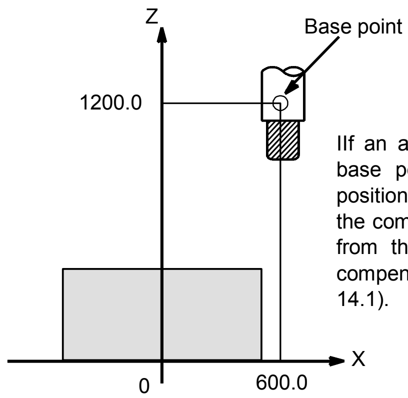
Example

M

(Example 1)
Setting the coordinate system by the G92X25.2Z23.0; command (The tool tip is the start point for the program.)



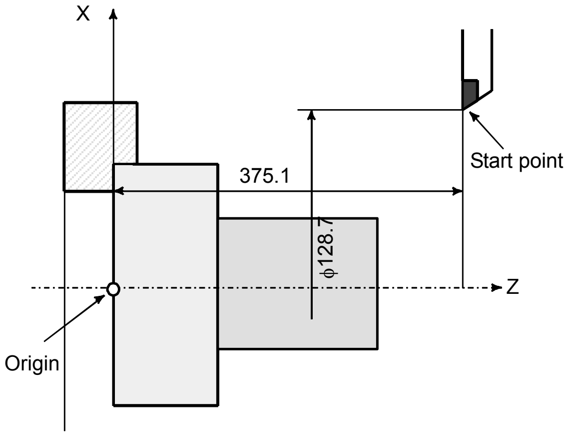
(Example 2)
Setting the coordinate system by the G92X600.0Z1200.0; command (The base point on the tool holder is the start point for the program.)



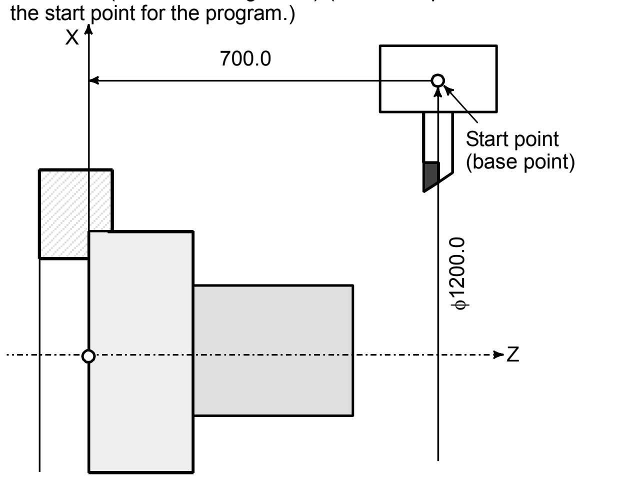
IIf an absolute command is issued, the base point moves to the commanded position. In order to move the tool tip to the commanded position, the difference from the tool tip to the base point is compensated by tool length offset (see II-14.1).

T

(Example 1)
Setting the coordinate system by the G50X128.7Z375.1; command (Diameter designation) (The tool nose is the start point for the program.)



(Example 2)
Setting the coordinate system by the G50X1200.0Z700.0; command (Diameter designation) (The base point on the turret is the start point for the program.)



> ⚠ **CAUTION**
> The set workpiece coordinate system depends on diameter programming or radius programming.

7.2.2 Selecting a Workpiece Coordinate System

The user can choose from set workpiece coordinate systems as described below. (For information about the methods of setting, see II-7.2.1.)

(1) Once a workpiece coordinate system is set by a workpiece coordinate system setting G code or by automatic workpiece coordinate system setting, absolute commands indicate positions in the workpiece coordinate system.

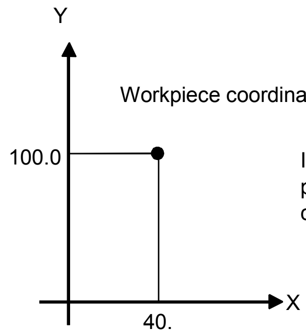
(2) Choosing from six workpiece coordinate systems set using the MDI panel
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.
G54 : Workpiece coordinate system 1
G55 : Workpiece coordinate system 2
G56 : Workpiece coordinate system 3
G57 : Workpiece coordinate system 4
G58 : Workpiece coordinate system 5
G59 : Workpiece coordinate system 6
Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

When bit 2 (G92) of parameter No. 1202 is set to 1, executing the workpiece coordinate system setting G92 code command results in the issue of an alarm PS0010. This is designed to prevent the user from confusing coordinate systems.

> ⚠ **CAUTION**
> The set workpiece origin offset value depends on diameter programming or radius programming.

Example

G90 G55 G00 X40.0 Y100.0 ;



In this example, positioning is made to positions (X=40.0, Y=100.0) in workpiece coordinate system 2.

Fig. 7.2.2 (a)

7.2.3 Changing Workpiece Coordinate System

The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece origin offset value or workpiece origin offset value.
Three methods are available to change an external workpiece origin offset value or workpiece origin offset value.

(1) Inputting from the MDI panel (see III-12.3.4)

(2) Programming (using a programmable data input G code or a workpiece coordinate system setting G code)

(3) Using the external data input function
An external workpiece origin offset value can be changed by input signal to CNC. Refer to machine tool builder's manual for details.

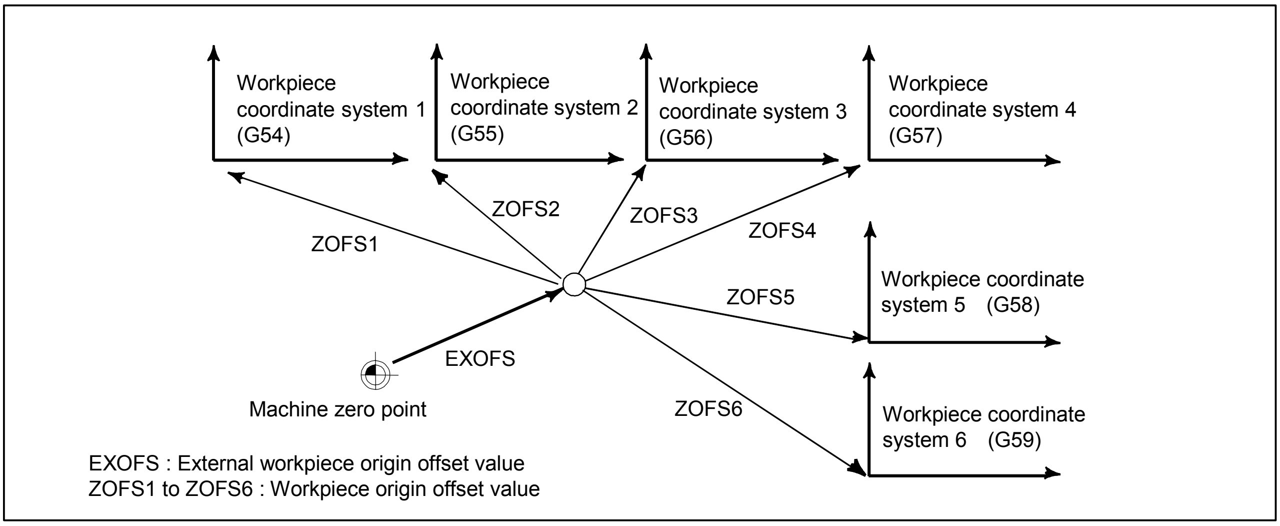


Fig. 7.2.3 (a) Changing an external workpiece origin offset value or workpiece origin offset value

Format

- Changing by inputting programmable data

G10 L2 Pp IP_;
p=0 : External workpiece origin offset value
p=1 to 6 : Workpiece origin offset value correspond to workpiece coordinate system 1 to 6
IP_ : For an absolute command, workpiece origin offset for each axis.
For an incremental command, value to be added to the set workpiece origin offset for each axis (the result of addition becomes the new workpiece origin offset).

- Changing by setting a workpiece coordinate system

M

G92 IP_ ;

T

G50 IP_ ;

Explanation

- Changing by inputting programmable data

By specifying a programmable data input G code, the workpiece origin offset value can be changed for each workpiece coordinate system.

- Changing by setting a workpiece coordinate system

By specifying a workpiece coordinate system setting G code, the workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).
Then, the amount of coordinate system shift is added to all the workpiece origin offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

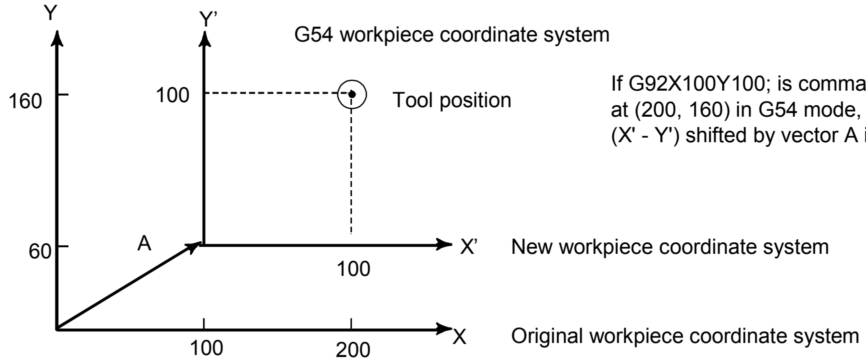
> ⚠ **CAUTION**
> When a coordinate system is set with workpiece coordinate system setting G92 code command after an external workpiece origin offset value is set, the coordinate system is not affected by the external workpiece origin offset value. When G92X100.0Z80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Z = 80.0 is set.

T

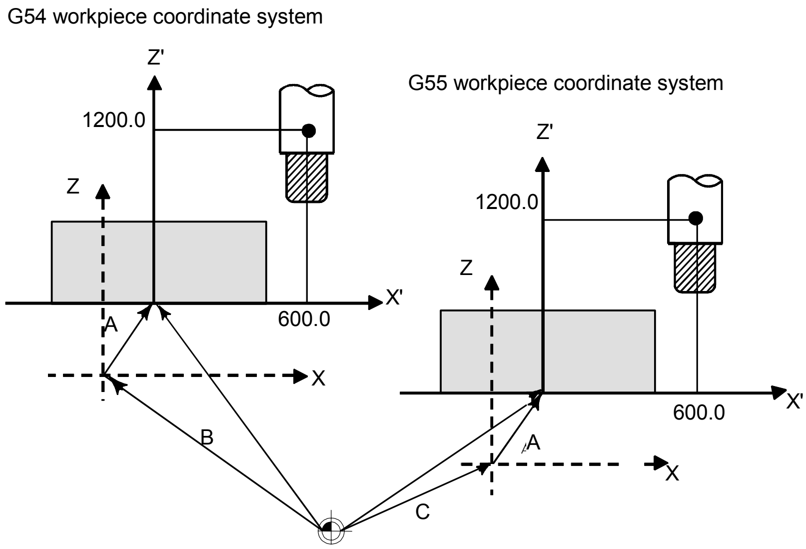
If IP is an incremental command value, the workpiece coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. (Coordinate system shift)

Example

M



If G92X100Y100; is commanded when the tool is positioned at (200, 160) in G54 mode, workpiece coordinate system 1 (X' - Y') shifted by vector A is created.



X' - Z' ....New workpiece coordinate system
X - Z ....Original workpiece coordinate system
A : Offset value created by G92
B : Workpiece origin offset value in the G54
C : Workpiece origin offset value in the G55

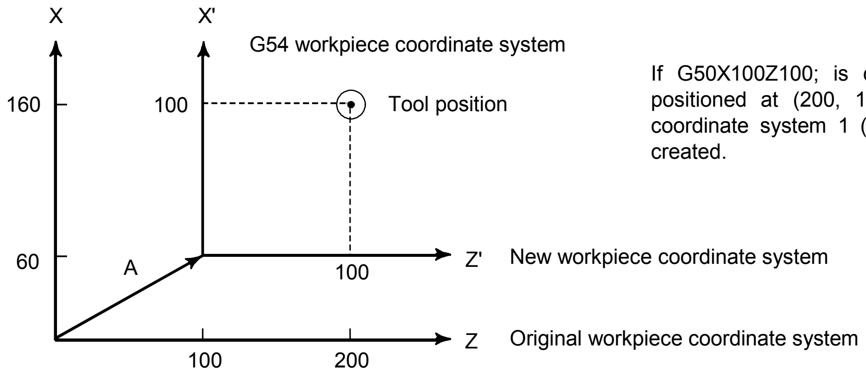
Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black circle on the tool (figure at the left) is at (600.0,1200.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:

G92X600.0Z1200.0;

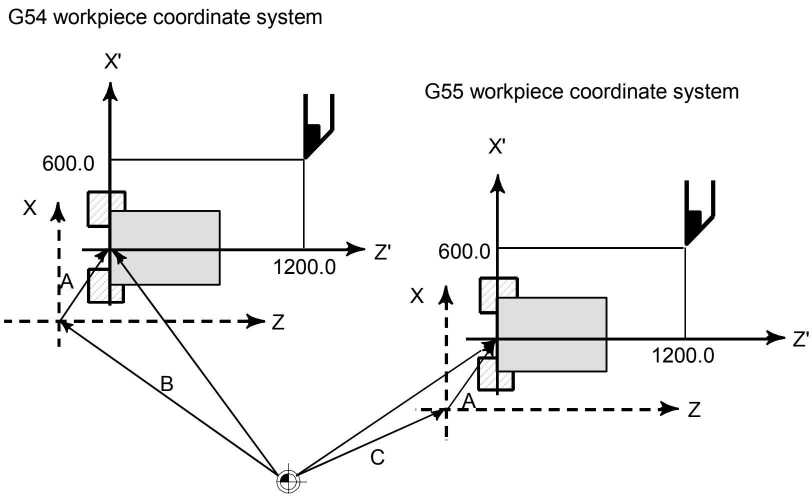
Also, suppose that pallets are loaded at two different positions. If the relative relationship of the coordinate systems of the pallets at the two positions is correctly set by handling the coordinate systems as the G54 workpiece coordinate system and G55 workpiece coordinate system, a coordinate system shift with G92 in one pallet causes the same coordinate system shift in the other pallet. This means that workpieces on two pallets can be machined with the same program just by specifying G54 or G55.

Example

T



If G50X100Z100; is commanded when the tool is positioned at (200, 160) in G54 mode, workpiece coordinate system 1 (X' - Z') shifted by vector A is created.



X' - Z'............ New workpiece coordinate system
X - Z .... Original workpiece coordinate system
A : Offset value created by G50
B : Workpiece origin offset value in G54
C : Workpiece origin offset value in G55

Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black tool nose point on the tool (figure at the left) is at (600.0,1200.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:

G50X600.0Z1200.0;

Also, suppose that loading is performed at two different places and that the G54 and G55 workpiece coordinate systems are set for these two places. If the relative relationship between the coordinate systems in the two places is correctly set, a coordinate system shift with G50 in one loading place causes the same coordinate system shift in the other loading place. This means that workpieces on the two loading positions can be machined with the same program just by specifying G54 or G55.

7.2.4 Workpiece Coordinate System Preset (G92.1)

The workpiece coordinate system preset function presets a workpiece coordinate system shifted by manual intervention to the pre-shift workpiece coordinate system. The latter system is displaced from the machine zero point by a workpiece origin offset value.
There are two methods for using the workpiece coordinate system preset function. One method uses a programmed command. The other uses MDI operations on the absolute position display screen, relative position display screen, and overall position display screen (see III-12.1.4).

Format

M

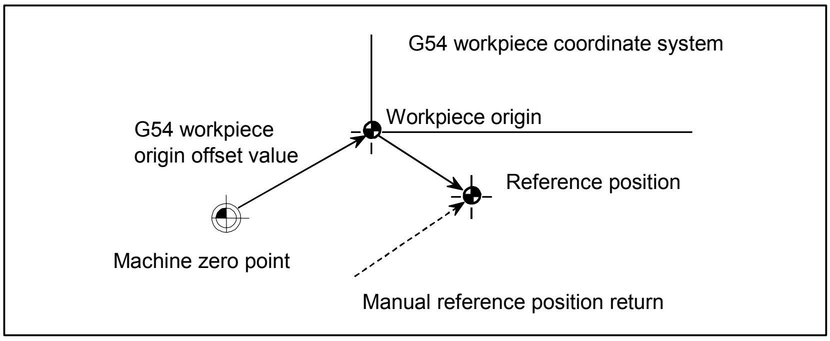
G92.1 IP 0 ;
IP 0 : Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

T

G92.1 IP 0; (G50.3 IP0 ; for G code system A)
IP 0 : Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

Explanation

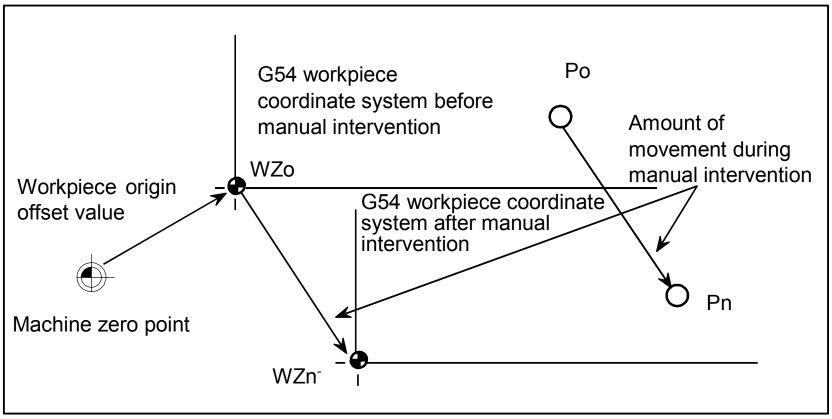
When manual reference position return operation is performed in the reset state, a workpiece coordinate system is shifted by the workpiece origin offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its origin displaced from the machine zero point by the G54 workpiece origin offset value; the distance from the origin of the workpiece coordinate system to the reference position represents the current position in the workpiece coordinate system.



If an absolute position detector is provided, the workpiece coordinate system automatically set at power-up has its origin displaced from the machine zero point by the G54 workpiece origin offset value. The machine position at the time of power-up is read from the absolute position detector and the current position in the workpiece coordinate system is set by subtracting the G54 workpiece origin offset value from this machine position. The workpiece coordinate system set by these operations is shifted from the machine coordinate system using the commands and operations listed below.

(a) Manual intervention performed when the manual absolute signal is off
(b) Move command executed in the machine lock state
(c) Movement by handle interruption
(d) Operation using the mirror image function
(e) Shifting the workpiece coordinate system by setting the local coordinate system or workpiece coordinate system

In the case of (a) above, the workpiece coordinate system is shifted by the amount of movement during manual intervention.



In the operation above, a workpiece coordinate system once shifted can be preset using G code (G92.1) specification or MDI operation to a workpiece coordinate system displaced by a workpiece origin offset value from the machine zero point.
Bit 3 (PPD) of parameter No. 3104 specifies whether to preset relative coordinates (RELATIVE) as well as absolute coordinates.
When no workpiece coordinate system option (G54 to G59) is selected, the workpiece coordinate system is preset to the coordinate system with its origin placed at the reference position.

Limitation

- Cutter or tool nose radius compensation, tool length compensation, tool offset

When using the workpiece coordinate system preset function, cancel compensation modes: cutter or tool nose radius compensation, tool length compensation, and tool offset. If the function is executed without canceling these modes, compensation vectors are cancelled.

M

- Tool length compensation

When using the workpiece coordinate system preset function, cancel tool length compensation. If the function is executed without canceling these modes, compensation vectors are cancelled.

- Program restart

The workpiece coordinate system preset function is not executed during program restart.

- Prohibited modes

Do not use the workpiece coordinate system preset function when the scaling, coordinate system rotation, programmable image, or figure copy mode is set.

7.2.5 Addition of Workpiece Coordinate System Pair (G54.1 or G54)

M

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 or 300 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

Format

- Selecting the additional workpiece coordinate systems

G54.1Pn ; or **G54Pn ;**
Pn : Codes specifying the additional workpiece coordinate systems
n : 1 to 48 or 1 to 300

- Setting the workpiece origin offset value in the additional workpiece coordinate systems (G10)

G10L20Pn IP_;
Pn : Codes specifying the workpiece coordinate system for setting the workpiece origin offset value
n : 1 to 48 or 1 to 300
IP_ : Axis addresses and a value set as the workpiece origin offset

Explanation

- Selecting the additional workpiece coordinate systems

When a P code is specified together with G54.1 (G54), the corresponding coordinate system is selected from the additional workpiece coordinate systems (1 to 48 or 1 to 300).
A workpiece coordinate system, once selected, is valid until another workpiece coordinate system is selected. Standard workpiece coordinate system 1 (selectable with G54) is selected at power-on.

G54.1 P1 .... Additional workpiece coordinate system 1
G54.1 P2 .... Additional workpiece coordinate system 2
:
G54.1 P48 .... Additional workpiece coordinate system 48
:
G54.1 P300 ..... Additional workpiece coordinate system 300

As with the standard workpiece coordinate systems, the following operations can be performed for a workpiece origin offset in an additional workpiece coordinate system:

(1) The workpiece origin offset value setting screen can be used to display and set a workpiece origin offset value.
(2) The G10 function enables a workpiece origin offset value to be set by programming (refer to II-7.2.3).
(3) A custom macro allows a workpiece origin offset value to be handled as a system variable.
(4) Workpiece origin offset data can be entered or output as external data.
(5) The PMC window function enables workpiece origin offset data to be read as program command modal data.

- Setting the workpiece origin offset value in the additional coordinate systems (G10)

When a workpiece origin offset value is specified using an absolute value, the specified value is the new offset value. When it is specified using an incremental value, the specified value is added to the current offset value to obtain a new offset value.

Limitation

- Specifying P codes

A P code must be specified after G54.1 (G54). If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1 (G54.1P1) is assumed.
If a value not within the specifiable range is specified in a P code, an alarm PS0030 is issued.
P codes other than workpiece offset numbers cannot be specified in a G54.1 (G54) block.

Example 1) G54.1G04P1000;
Example 2) G54.1M98P48;

7.2.6 Automatic Coordinate System Setting

When ZPR (bit 0 of parameter No. 1201) for automatic coordinate system setting is 1, a coordinate system is automatically determined when manual reference position return is performed.
Once α, β, and γ are set with parameter No. 1250, a workpiece coordinate system is set upon reference position return so that the base point on the tool holder or the tip of the basic tool is positioned at X = α, Y = β, and Z = γ.
This processing occurs as if the following are specified at the reference position:

M

G92X$\underline{\alpha}$Y$\underline{\beta}$Z$\underline{\gamma}$;

T

G50X$\underline{\alpha}$Z$\underline{\gamma}$;

This function cannot be used, however, when the workpiece coordinate system option is selected.

T

When the setting of a workpiece coordinate system shift amount is other than 0, a workpiece coordinate system shifted by the amount is set.

7.2.7 Workpiece Coordinate System Shift

T

Explanation

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed workpiece system, the set coordinate system can be shifted (see III-3.1).
Set the desired shift amount in the workpiece coordinate system shift memory.

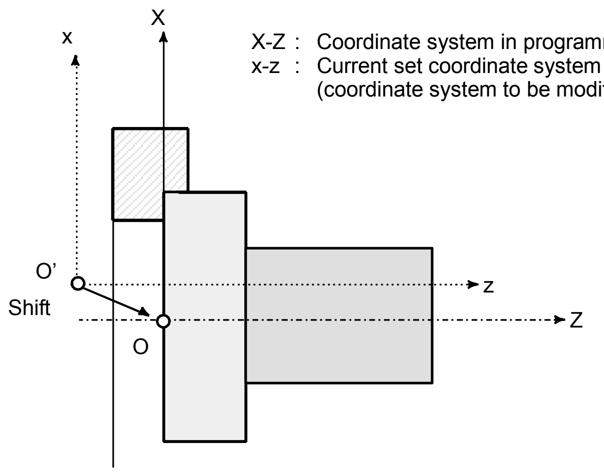


Fig. 7.2.7 (a) Workpiece coordinate system shift

Format

- Changing the workpiece coordinate system shift amount

G10 P0 IP_;

IP : Settings of an axis address and a workpiece coordinate system shift amount

> ⚠ **CAUTION**
> A single block can contain a combination of X, Y, Z, C, U, V, W, and H (in G code system A). In this case, if commands are specified for the same axis, whichever appears later becomes valid.

Limitation

- Shift amount and coordinate system setting command

Specifying a coordinate system setting command (G50 or G92) invalidates the shift amount that has already been set.

Example) When G50X100.0Z80.0; is specified, a coordinate system is set so that the current base position of the tool is at X =100.0 and Z = 80.0, regardless of which value has been set for the workpiece coordinate system shift amount.

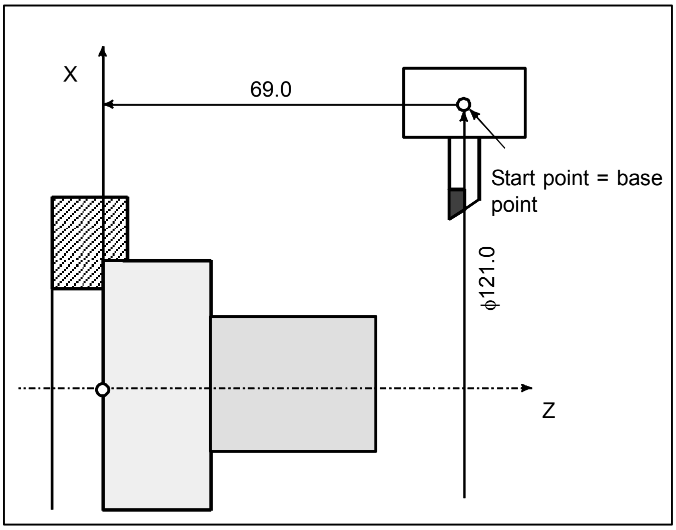
- Shift amount and coordinate system setting

After a shift amount is set, when automatic coordinate system setting is performed upon manual reference position return, the set coordinate system is immediately shifted by the set amount.

- Diameter and radius values

The workpiece coordinate system shift amount depends on diameter programming or radius programming.

Example) Although the base point should be positioned at X = ϕ120.0 (diameter value) and Z = 70.0 from the workpiece origin, the actual position is at X = ϕ121.0 and Z = 69.0 from the origin. Set a shift amount as shown below: X=1.0, Z=-1.0



7.3 LOCAL COORDINATE SYSTEM

When a program is created in a workpiece coordinate system, a child workpiece coordinate system can be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format

G52 IP_;	Setting the local coordinate system
:	
G52 IP 0 ;	Canceling of the local coordinate system
IP_ : Origin of the local coordinate system	

Explanation

By specifying G52 IP_;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system.
Once a local coordinate system is established, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the origin of a new local coordinate system in the workpiece coordinate system.
To cancel the local coordinate system or specify the coordinate value in the workpiece coordinate system, match the origin of the local coordinate system with that of the workpiece coordinate system.

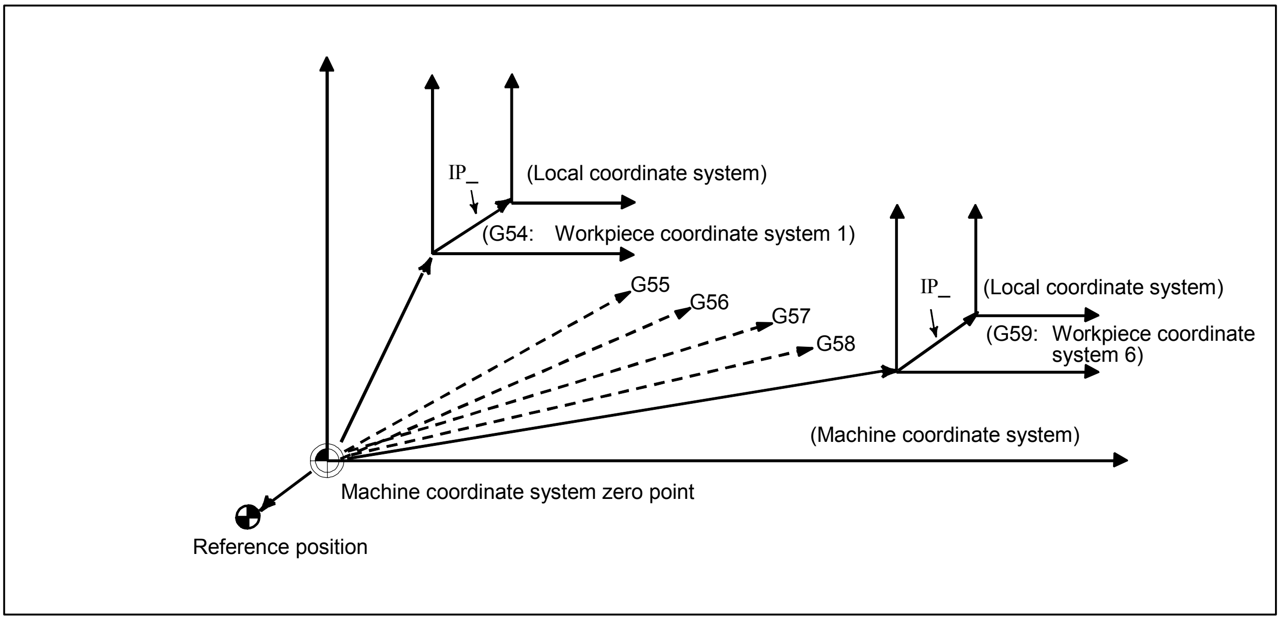


Fig. 7.3 (a) Setting the local coordinate system

⚠ CAUTION

1 When ZCL (bit 2 of parameter No.1201) is set to 1 and an axis returns to the reference position by the manual reference position return function, the origin of the local coordinate system of the axis matches that of the workpiece coordinate system. The same is true when the following command is issued:

  G52α0;

   α: Axis which returns to the reference position

2 The local coordinate system setting does not change the workpiece and machine coordinate systems.

3 Whether the local coordinate system is canceled at reset depends on the parameter setting. The local coordinate system is canceled when either CLR, bit 6 of parameter No.3402 or RLC, bit 3 of parameter No.1202 is set to 1. In three-dimensional coordinate conversion mode, however, the local coordinate system is not canceled when D3R (bit 2 of parameter No. 5400) is set to 1.

4 When a workpiece coordinate system is set with the G92 (G50 for G code system A in the T series) command, the local coordinate system is canceled. However, the local coordinate system of an axis for which no coordinate system is specified in a G92 (G50 for G code system A in the T series) block remains unchanged.

5 G52 cancels the offset temporarily in cutter or tool nose radius compensation.

6 Command a move command immediately after the G52 block in the absolute mode.

7.4 PLANE SELECTION

Select the planes for circular interpolation, cutter compensation, and drilling by G-code.
The following table lists G-codes and the planes selected by them.

Explanation

Table 7.4 (a) Plane selected by G code

G code	Selected plane	Xp	Yp	Zp
G17	Xp Yp plane	X-axis or an axis parallel to it	Y-axis or an axis parallel to it	Z-axis or an axis parallel to it
G18	Zp Xp plane			
G19	Yp Zp plane			

Xp, Yp, Zp are determined by the axis address appeared in the block in which G17, G18 or G19 is commanded.
When an axis address is omitted in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.
Parameter No. 1022 is used to specify that an optional axis be parallel to the each axis of the X-, Y-, and Z-axes as the basic three axes.
The plane is unchanged in the block in which G17, G18 or G19 is not commanded.
The movement instruction is irrelevant to the plane selection.

M

When the power is turned on or the CNC is reset, G17 (XY plane), G18 (ZX plane), or G19 (YZ plane) is selected by of parameters G18 and G19 (No. 3402#1 and #2).

T

When the power is turned on, G18 (ZX plane) is selected .

> **NOTE**
> 1 U-, V-, and W-axes can be used with G-codes B and C.
> 2 Direct drawing dimension programming, chamfering, corner R, multiple repetitive canned cycle, and simple canned cycle are enabled only for the ZX plane. Specifying these functions for other planes causes alarm PS0212 to be generated.

Example

Plane selection when the X-axis is parallel with the U-axis.

```
G17X_Y_     XY plane,
G17U_Y_     UY plane
G18X_Z_     ZX plane
   X_Y_     Plane is unchanged (ZX plane)
G17         XY plane
G18         ZX plane
G17 U_      UY plane
G18Y_ ;     ZX plane, Y axis moves regardless without any
            relation to the plane.
```

8 COORDINATE VALUE AND DIMENSION

This chapter contains the following topics.

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING

There are two ways to command travels of the tool; the absolute command, and the incremental command. In the absolute command, coordinate value of the end position is programmed. The incremental command is used to program the amount of a tool movement.

M

G90 and G91 are used to command absolute or incremental command, respectively.

T

Absolute programming or incremental programming is used depending on the command used. See following tables.

G code system	A	B or C
Command method	Address word	G90, G91

Format

M

Absolute command	**G90 IP_ ;**
Incremental command	**G91 IP_ ;**

T

- G code system A

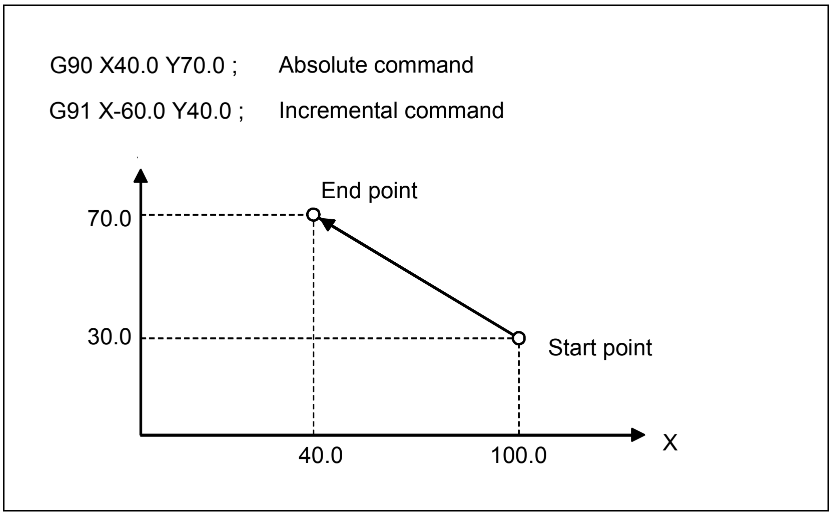
	Absolute command	Incremental command
X axis move command	X	U
Z axis move command	Z	W
Y axis move command	Y	V
C axis move command	C	H

- G code system B or C

Absolute command	**G90 IP_ ;**
Incremental command	**G91 IP_ ;**

Example

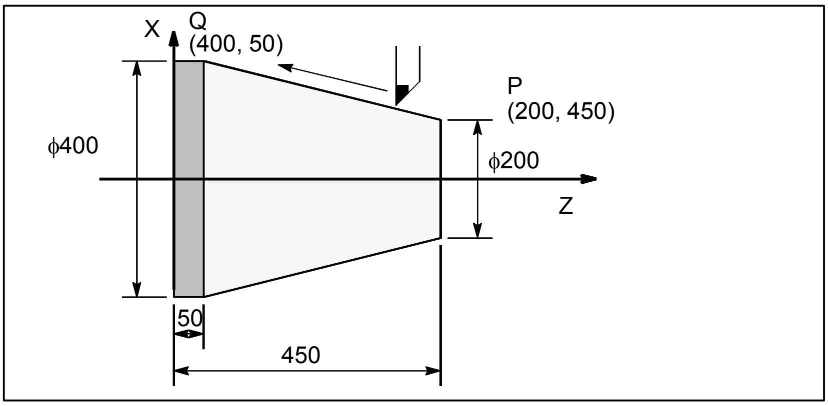
M



T

Tool movement from point P to point Q (diameter programming is used for the X-axis)

	G code system A	G code system B or C
Absolute command	X400.0 Z50.0 ;	G90 X400.0 Z50.0 ;
Incremental command	U200.0 W-400.0 ;	G91 X200.0 Z-400.0 ;



NOTE

1 Absolute and incremental commands can be used together in a block.
In the above example, the following command can be specified : X400.0 W-400.0 ; (in the G code system A)

2 When both X and U or W and Z are used together in a block, the one specified later is effective.

3 Incremental commands cannot be used when names of the axes are A and B during G code system A is selected.

8.2 INCH/METRIC CONVERSION (G20, G21)

Either inch or metric input (least input increment) can be selected by G code.

Format

G20 ;	Inch input
G21 ;	Metric input

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. After the G code for inch/metric conversion is specified, the unit of input data is switched to the least inch or metric input increment of increment system (II-2.3). The unit of data input for degrees remains unchanged. The unit systems for the following values are changed after inch/metric conversion:

- Feedrate commanded by F code
- Positional command
- Workpiece origin offset value
- Tool compensation value
- Unit of scale for manual pulse generator
- Movement distance in incremental feed
- Some parameters

When the power is turned on, the G code is the same as that held before the power was turned off.

⚠ **WARNING**

G20 and G21 must not be switched during a program.

NOTE

1 When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.

2 The inch and metric input can also be switched using settings (see III-12.3.1).

8.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:

M

X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, F

T

X, Y, Z, U, V, W, A, B, C, I, J, K, R, F

Explanation

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal notation is used, a value without decimal point is considered to be specified in millimeters inch,or deg. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal notation by using the parameter DPI (No. 3401#0).Values can be specified both with and without decimal point in a single program.

Example

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit :mm	1mm Unit : Least input increment (0.001mm)
X1000.0 Command value with decimal point	1000mm Unit :mm	1000mm Unit :mm

> ⚠ **CAUTION**
> When specifying a dimension word for a command G code in a block, be sure to place the dimension word after the command G code.

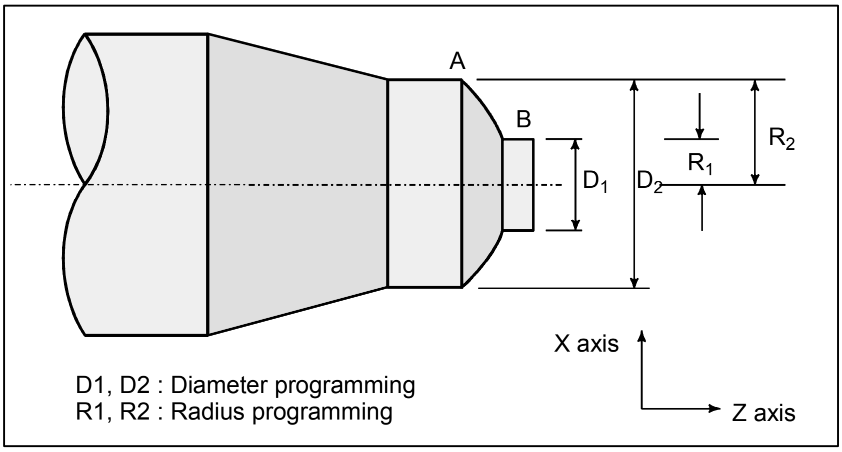
NOTE

1 A specified value less than the minimum unit is treated as described below.
Example 1)
When a value is specified directly at an address (in the case of IS-B)
X1.2345 ; Treated as X1.235
X-1.2345 ; Treated as X-1.234
Example 2)
When a value is assigned to a macro variable (in the case of IS-B)
#100=1.2345;
X#100 ; Treated as X1.235
#100=-1.2345;
X#100 ; Treated as X-1.234

2 When more than eight digits are specified, an alarm occurs. If a value is entered with a decimal point, the number of digits is also checked after the value is converted to an integer according to the least input increment.
Examples:
X1.23456789;
Alarm PS0003 occurs because more than eight digits are specified.
X123456.7;
If the least input increment is 0.001 mm, the value is converted to integer 123456700. Because the integer has more than eight digits, an alarm occurs.

8.4 DIAMETER AND RADIUS PROGRAMMING

Since the workpiece cross section is usually circular in CNC lathe control programming, its dimensions can be specified in two ways :
Diameter and Radius



When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming.

Explanation

- Notes on diameter programming/radius programming for each command

Radius programming or diameter programming can be specified by parameter DIA (No.1006#3). When using diameter programming, note the conditions listed in the Table 8.4 (a).

Table 8.4 (a) Notes on specifying diameter value

Item	Notes
X axis command	Specified with a diameter value
Incremental command	Specified with a diameter value In the above figure, specifies D2 minus D1 for tool path B to A.
Coordinate system setting (G50)	Specifies a coordinate value with a diameter value
Component of tool offset value	Parameter (No.5004#1) determines either diameter or radius value
Parameters in canned cycle, such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation (R, I, K, and etc.)	Specifies a radius value
Feedrate along axis	Specifies change of radius/rev. or change of radius/min.
Display of axis position	Displayed as diameter value

8.5 DIAMETER AND RADIUS SETTING SWITCHING FUNCTION

Overview

Usually, whether to use diameter specification or radius specification to specify a travel distance on each axis is uniquely determined by the setting of bit 3 (DIAx) of parameter No. 1006. However, this function enables switching between diameter specification and radius specification by using a signal or G code.
Thus, a coordinate, program, and so forth can be specified by switching between diameter specification and radius specification for each controlled axis.

Explanation

- Selection of a diameter/radius specification switching method

Two methods are available for switching between diameter specification and radius specification:
1) Signal
2) G code
Use bit 5 (PGD) of parameter No. 3400 to determine which method to use.

- Switching method using a signal

For switching between diameter specification and radius specification, set the diameter/radius specification switch signal from DI1 to DI8 (input signals) corresponding to a desired axis, from 0 to 1.
If an input signal is set from 0 to 1, and radius specification is selected (with bit 3 (DIAx) of parameter No. 1006 = 0) for the axis corresponding to the input signal, the specification method switches to diameter specification; the specification method switches to radius specification if diameter specification is selected (with bit 3 (DIAx) of parameter No. 1006 = 1).
During switching, the diameter/radius specification switching in-progress signal from DM1 to DM8 (output signals) corresponding to a switched axis is output.
To return the diameter/radius specification of an axis to the original state, set the setting of the corresponding diameter/radius specification switch signal from DI1 to DI8, from 1 to 0.

NOTE

1 When operating an input signal by using an M code, for example, during automatic operation, perform a switching operation according to the method below to reflect the state of diameter/radius specification switching in the execution block correctly.
As an auxiliary function for switching, use an unbuffered M code (parameter No. 3411 and up).
Use the following sequences for a specified M code:
- When switching is performed
  M code → Input signal ON → Confirmation of output signal ON → FIN
- When switching is cancelled
  M code → Input signal OFF → Confirmation of output signal OFF → FIN

If a diameter/radius specification switch signal is operated during automatic operation without following the sequences above, alarm PS5320 is issued.

2 If a diameter/radius specification switch signal is operated while a movement is made on an axis subject to switching, alarm PS5320 is issued.

- Switching method using a G code (programmable diameter/radius specification switching)

The format of a G code for diameter/radius specification switching is as follows:

Format

G10.9 IP_ ;

IP_ : Address and command value of an specified axis subject to diameter/radius specification switching
Specify 0 or 1 as the command value.
0: Radius specification
1: Diameter specification

NOTE

1 Specify G10.9 in a single block specifying no other codes.
2 After an axis address, specify a command value without using the decimal point.

- Switching operation

According to the switching methods above, diameter/radius specification is internally switched as described below.

1) Switching using a signal
   - When parameter DIAx = 0 (radius specification) → Operation is performed with diameter specification.
   - When parameter DIAx = 1 (diameter specification) → Operation is performed with radius specification.

2) Switching using a G code
   - When the specified address value = 0 (radius specification) → Operation is performed with radius specification.
   - When the specified address value = 1 (diameter specification) → Operation is performed with diameter specification.

NOTE

1 When the diameter/radius specification switching state is to be cancelled using a reset or mode switching at the time of signal-based switching, the input signal needs to be operated.

2 Switching using a G code is cancelled by a reset.

⚠ CAUTION

When switching is performed from diameter specification to radius specification, the travel distance based on the same move command is doubled when compared with diameter specification. So, when switching from diameter specification to radius specification, ensure safety in machine operation.

Limitation

- Feedrate

A radius-based feedrate is specified in both of diameter specification and radius specification at all times.

- Data not switchable

The following data follows the setting of parameter DIAx, so that diameter/radius specification switching is not performed:
- Parameter
- Offset
- Workpiece coordinate system
- Scale display on the graphic screen

NOTE

For offset data, the settings of bit 1 (ORC) of parameter No. 5004 and bit 2 (ODI) of parameter No. 5004 have priority.

- Switchable data and commands

For the following data and commands, diameter/radius specification switching is performed according to the specified specification method:

- Programmed move command
- Current position display
- Workpiece coordinate system preset
- Movement based on the manual numeric command G00 or G01

- Use with other functions

1) Diameter/radius specification switching cannot be performed for an axis on which a movement is being made with any of the functions indicated below.
   Moreover, none of the functions indicated below can be performed during diameter/radius specification switching.
   - Synchronous/mixture control
   - Superimposed Control
   - Axis synchronous control
   - PMC axis control

2) This function is ignored and inoperative when the following function is performed:
   - Background graphic

9 SPINDLE SPEED FUNCTION (S FUNCTION)

The spindle speed can be controlled by specifying a value following address S.
This chapter contains the following topics.

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE

When a value is specified after address S, the code signal and strobe signal are sent to the machine to control the spindle rotation speed.
A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the number of digits in an S code or the execution order when a move command and an S code command are in the same block.

9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND)

The spindle speed can be specified directly by address S followed by a max. five-digit value (min^{-1}). The unit for specifying the spindle speed may vary depending on the machine tool builder. Refer to the appropriate manual provided by the machine tool builder for details.

9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)

Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the position of the tool.

Format

- Constant surface speed control command

G96Sxxxxx ;
↑ Surface speed (m/min or feet/min)
This surface speed unit may change according to machine tool builder's specification.

- Constant surface speed control cancel command

G97Sxxxxx ;
↑ Spindle speed (min^{-1})
This surface speed unit may change according to machine tool builder's specification.

- Constant surface speed controlled axis command

G96Pα ;
P0 : Axis set in the parameter (No. 3770)
P1 : X axis, P2 : Y axis, P3 : Z axis, P4 : 4th axis
P5 : 5th axis, P6 : 6th axis, P7 : 7th axis, P8 : 8th axis

- Clamp of maximum spindle speed

G92 S_ ;
The maximum spindle speed (min^{-1}) follows S.

T

G50 S_ ;
The maximum spindle speed (min^{-1}) follows S.

NOTE
G50 can be used with G code system A.

Explanation

- Constant surface speed control command (G96)

G96 (constant surface speed control command) is a modal G code. After a G96 command is specified, the program enters the constant surface speed control mode (G96 mode) and specified S values are assumed as a surface speed. A G96 command must specify the axis along which constant surface speed control is applied. A G97 command cancels the G96 mode. When constant surface speed control is applied, a spindle speed higher than the value specified in G92S_; or G50S_; (maximum spindle speed) is clamped at the maximum spindle speed. When the power is turned on, the maximum spindle speed is not yet set and the speed is not clamped. S (surface speed) commands in the G96 mode are assumed as S = 0 (the surface speed is 0) until M03 (rotating the spindle in the positive direction) or M04 (rotating the spindle in the negative direction) appears in the program.

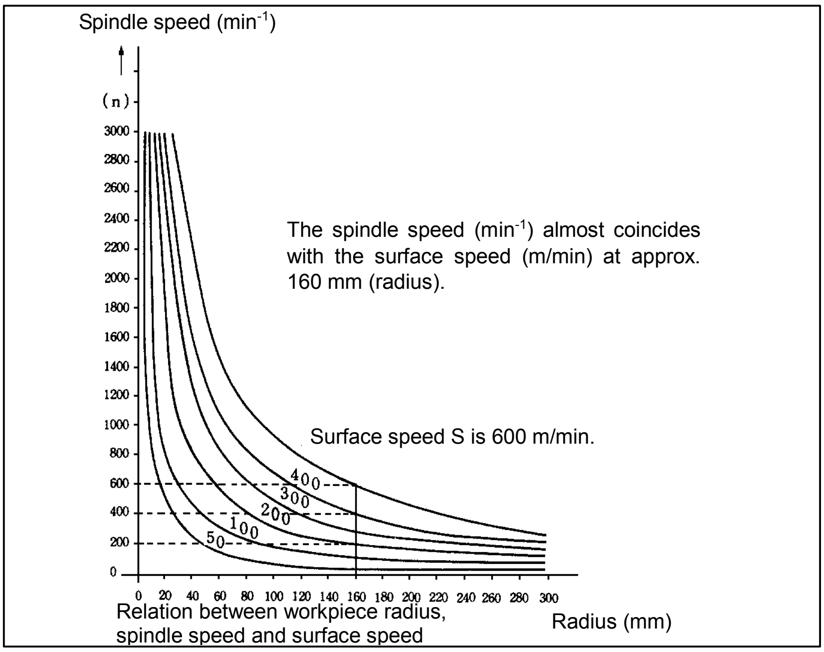


Fig. 9.3 (a) Relation between workpiece radius, spindle speed and surface speed

- Setting the workpiece coordinate system for constant surface speed control

To execute the constant surface speed control, it is necessary to set the workpiece coordinate system , and so the coordinate value at the center of the rotary axis, for example, Z axis, (axis to which the constant surface speed control applies) becomes zero.

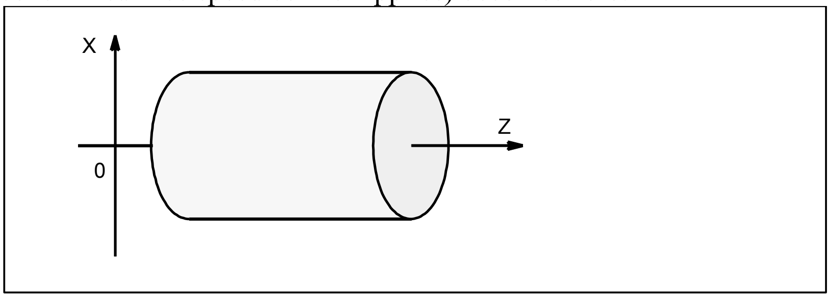
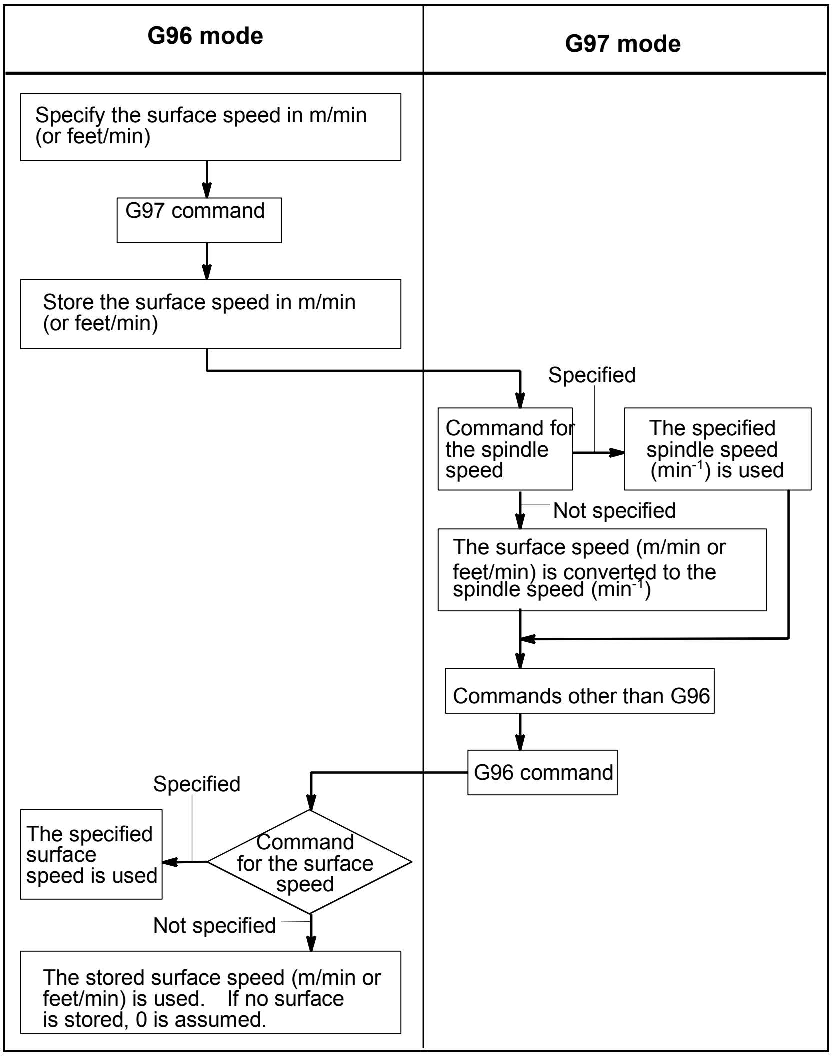


Fig. 9.3 (b) Example of the workpiece coordinate system for constant surface speed control

- Surface speed specified in the G96 mode



Limitation

- Constant surface speed control for threading

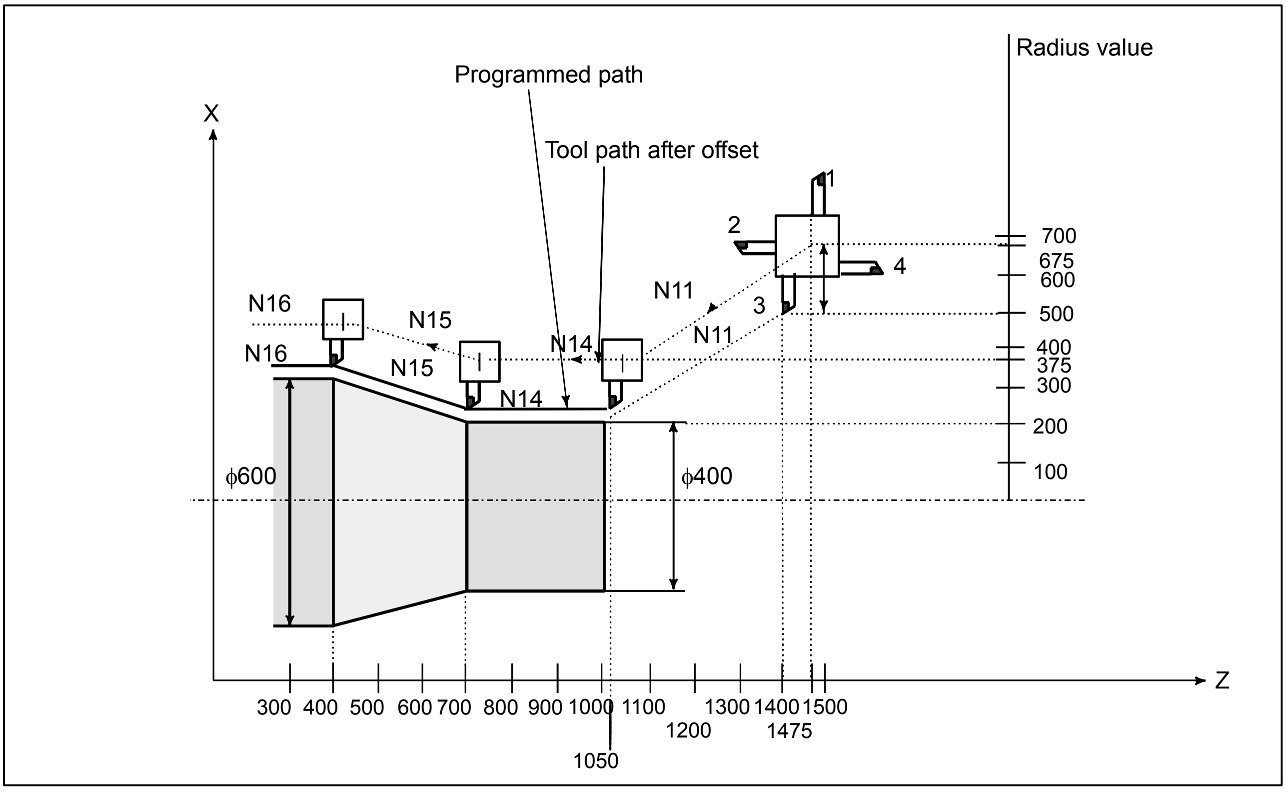
The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.

- Constant surface speed control for rapid traverse (G00)

In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block, on the condition that cutting is not executed at rapid traverse.

Example

T



N8 G00 X1000.0Z1400.0 ;
N9 T33;
N11 X400.0Z1050.0;
N12 G50S3000 ; (Designation of max. spindle speed)
N13 G96S200 ; (Surface speed 200 m/min)
N14 G01 Z 700.0F1000 ;
N15 X600.0Z 400.0;
N16 Z_ ;

The CNC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis. This is not the value calculated according to the X axis coordinate after offset when offset is valid. At the end point N15 in the example above, the speed at 600 dia. (Which is not the turret center but the tool nose) is 200 m/min. If X axis coordinate value is negative, the CNC uses the absolute value.

9.4 SPINDLE POSITIONING FUNCTION

Overview

In turning, the spindle connected to the spindle motor is rotated at a certain speed to rotate the workpiece mounted on the spindle. This spindle control status is referred to as spindle rotation mode.
The spindle positioning function turns the spindle connected to the spindle motor by a certain angle to position the workpiece mounted on the spindle at a certain angle. This spindle control status is referred to as spindle positioning mode.

The spindle positioning function involves the following three operations:

1. Canceling the spindle rotation mode and entering the spindle positioning mode
   Place the spindle in the spindle positioning mode and establish a reference position by specifying a given M code (set with a parameter). (Spindle orientation)
2. Positioning the spindle in the spindle positioning mode
   The spindle is positioned in either of the two methods:
   1) Positioning with an arbitrary angle by an axis address
   2) Positioning with a semi-fixed angle by a given M code (set with a parameter)
3. Canceling the spindle positioning mode, and entering the spindle rotation mode
   Place the spindle in the spindle rotation mode by specifying a given M code (set with a parameter).

The least command increment, least input increment, and maximum value for the spindle positioning axis are as follows:

- Least command increment
  $\frac{360}{4096} \fallingdotseq 0.088$ deg (when the gear ratio of the spindle to the position coder
- Least input increment
  0.001 deg (IS-B)
- Maximum value
  ±999999.999 deg

9.4.1 Spindle Orientation

When spindle positioning is first performed after the spindle motor is used for normal spindle operation, or when spindle positioning is interrupted, the spindle orientation is required.
Orientation permits the spindle to stop at a predetermined position.
Orientation is directed by the M code set in parameter No. 4960. The direction of orientation can be set with a parameter.
For the serial spindle, it is set in RETSV (bit 4 of parameter No. 4000).
With the grid shift function, the orientation position can be shifted in a range of 0 to 360 deg with parameter No. 4073 for a serial spindle.

- Feedrate during spindle orientation

An orientation feedrate for a serial spindle is determined by a spindle parameter setting.
In orientation, the serial spindle stops at the orientation position after several rotations of the spindle motor.

- Omission of orientation

By using ISZ (bit 2 of parameter No. 4950), orientation upon switching to spindle positioning mode can be omitted if it is unnecessary (for example, when no start position is specified and incremental positioning from the current position is only required). More specifically, when an M code for switching to spindle positioning mode is specified, the spindle control mode is simply switched to spindle positioning mode and then the processing is completed without orientation.

- Program reference position

The position at which orientation is completed is assumed to be a program reference position. However, the program reference position can be changed through coordinate system setting (G92 or G50) or automatic coordinate system setting (ZPR (bit 0 of parameter No. 1201)).
When a setting is made to omit orientation, a program reference position is not established, and operation by an absolute command is unpredictable during spindle positioning with an axis address.

9.4.2 Spindle Positioning

The spindle can be positioned with a semi-fixed angle or arbitrary angle.

- Positioning with a semi-fixed angle

Use an M code to specify a positioning angle. The specifiable M code value may be one of the six values from $M\alpha$ to $M(\alpha+5)$. Value α must be set in parameter No. 4962 beforehand. The positioning angles corresponding to $M\alpha$ to $M(\alpha+5)$ are listed below. Value β must be set in parameter No. 4963 beforehand.

M-code (Ex.) $\beta = \alpha + 5$	Positioning angle	(Ex.) β = 30°
$M\alpha$	β	30°
$M(\alpha + 1)$	2β	60°
$M(\alpha + 2)$	3β	90°
$M(\alpha + 3)$	4β	120°
$M(\alpha + 4)$	5β	150°
$M(\alpha + 5)$	6β	180°

When the number of M codes to be used, value γ, is specified in parameter No. 4964, a specifiable M code value may be in a range of values from $M\alpha$ to $M(\alpha + (\gamma - 1))$, up to 255 values from $M\alpha$ to $M(\alpha + (255 - 1))$.

M code (Ex.) $\gamma = 11$	Positioning angle	(Ex.) β = 30°
$M\alpha$	β	30°
$M(\alpha + 1)$	2β	60°
$M(\alpha + 2)$	3β	90°
$M(\alpha + 3)$	4β	120°
...	...	...
$M(\alpha + 11 - 1)$	11β	330°

The direction of rotation can be specified in IDM (bit 1 of parameter 4950).

- Positioning with an arbitrary angle

Specify the position with an arbitrary angle using the axis address followed by a signed numeric value or numeric values. The axis address must be specified in the G00 mode.
(The explanation below is given assuming that a C axis address is set.)

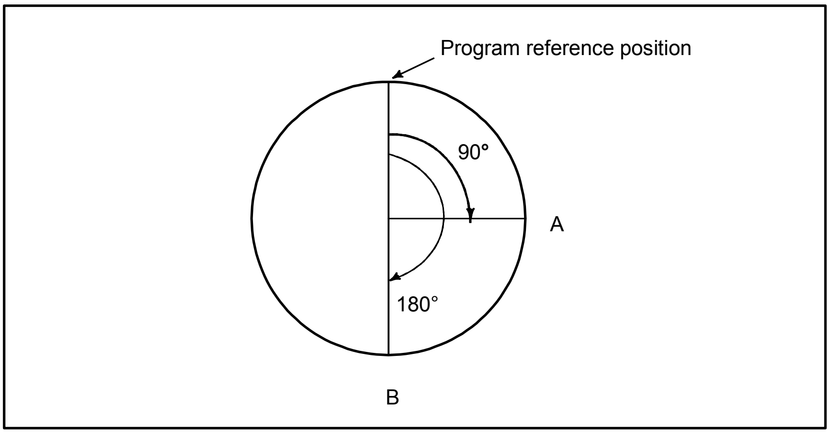
(Example) C-45000
C180.000

A numeric with the decimal point can be entered. The value must be specified in degrees.

(Example) C36.0=C36 degrees

- Absolute commands and incremental commands

Incremental commands are always used for positioning with a semi-fixed angle (using M codes).
The direction of rotation can be specified with IDM (bit 1 of parameter No. 4950).
Absolute and incremental commands can be used for positioning with an arbitrary angle.
With absolute commands for positioning with an arbitrary angle, when the rotation axis rollover function is used (ROA (bit 0 of parameter No. 1008) is 1), shortcut control is also enabled (RAB (bit 1 of parameter No. 1008) is 0).



Command format		G-code system A in lathe system		G-code system B or C in lathe system, and machining center system	
		Address used	Command A-B in the above figure	Address used and G code	Command A-B in the above figure
Absolute command	Specify the end point with a distance from the program reference position.	C	C180.0 ;	G90, C	G90 C180 .;
Incremental command	Specify a distance from the start point to the end point.	H	H90.0 ;	G91, C	G90 C90 .;

- Feedrate during positioning

The feedrate during positioning equals the rapid traverse speed specified in parameter No. 1420.
For the specified speed, an override of 100%, 50%, 25%, and F0 (parameter No. 1421) can be applied.

9.4.3 Canceling Spindle Positioning

When modes are to be switched from spindle positioning to normal spindle rotation, the M code set in parameter No. 4961 must be specified.

Also, the spindle positioning mode is canceled and the spindle rotation mode is set in the following cases:

<1> A reset operation (including an emergency stop) occurs when a servo alarm is issued.

<2> A reset operation (including an emergency stop) occurs when a spindle alarm is issued.

<3> An orientation operation in progress is stopped due to a reset or alarm, or for some other reason.

<4> A reset operation (including an emergency stop) occurs when IOR (bit 0 of parameter No. 4950) is 1.

> ⚠ **CAUTION**
> 1 During execution of spindle positioning sequences (canceling the spindle rotation mode and entering the spindle positioning mode, positioning the spindle in the spindle positioning mode, and canceling the spindle positioning mode and entering the spindle rotation mode), the automatic operation stop signal *SP is invalid. This means that automatic operation does not stop until all the sequences are completed, even when the *SP signal becomes 0.
> 2 Dry run and machine lock cannot be performed during spindle positioning.
> 3 Auxiliary function lock is disabled for M codes for the spindle positioning function.
> 4 The serial spindle Cs-axis contour control function and the spindle positioning function cannot be used at a time. If both options are specified, the spindle positioning function has priority.
> 5 The spindle positioning axis is handled as a controlled axis. Therefore, controlled axis-related signals (such as the overtravel signal) must be set.
> 6 When using the rigid tapping function and the spindle positioning function concurrently, do not specify rigid tapping in spindle positioning mode or spindle positioning in rigid tapping mode.

NOTE

1 M code commands for positioning of a spindle must be specified in a single block. Other commands must not be contained in the same block. (Also, M code commands for positioning of another spindle must not be contained in the same block.)
Even when the single-block, multiple-M code command function is also used, related M codes must be specified in a single block.

2 Even when the single-block, multiple-M code command function is also used, related M codes must be specified in a single block.

3 Axis address commands for positioning of a spindle must be specified in a single block. Other commands must not be contained in the same block. However, the following commands can be contained in the same block where axis address commands are specified:
G00, G90, G91, G92 (G-code systems B and C in the T series, M series)
G00, G50 (G-code system A in the T series)

4 M code commands for spindle positioning specify M codes that are not buffered.

5 Spindle positioning cannot be performed by manual operation (in jog feed, manual handle feed, manual numeric command, or other mode).

6 Spindle positioning cannot be performed by PMC axis control.

7 For spindle positioning, program restart and block restart operations cannot be performed. Use the MDI for these operations.

8 The stored stroke limit check is disabled for the spindle positioning axis.

9 The axis removal function is disabled for the spindle positioning axis.

10 The pitch error compensation function is disabled for the spindle positioning axis.

11 When a setting is made to omit spindle orientation, the reference position return completion signal does not become 1.

12 In spindle orientation, all-axis interlock and axis-specific interlock are checked only when a block is started. A signal is ignored if input during the execution of the block.

13 A difference between a specified travel distance and an actual travel distance is maintained until spindle positioning mode is canceled.

9.5 SPINDLE SPEED FLUCTUATION DETECTION

Overview

With this function, an overheat alarm (OH0704) is raised and the spindle speed fluctuation detection alarm signal SPAL is issued when the spindle speed deviates from the specified speed due to machine conditions.
This function is useful, for example, for preventing the seizure of the guide bushing.
G26 enables spindle speed fluctuation detection.
G25 disables spindle speed fluctuation detection.

Format

- Spindle fluctuation detection on

G26 Pp Qq Rr Ii ;

P: Time (in ms) from the issue of a new spindle rotation command (S command) to the start of checking whether the actual spindle speed is so fast that an overheat can occur.
When a specified speed is reached within the time period of P, a check is started at that time.

Q: Tolerance (%) of a specified spindle speed

$$q = \left| \frac{1 - actual\ spindle\ speed}{specified\ spndle\ speed} \right| \times 100$$

If a specified spindle speed lies within this range, it is regarded as having reached the specified value. Then, the checking of an actual spindle speed is started.

R: Spindle speed fluctuation (%) at which the actual spindle speed is so fast that an overheat can occur

$$r = \left| \frac{1 - speed\ that\ can\ cause\ overheat}{speciofied\ spndle\ speed} \right| \times 100$$

If the fluctuation of the actual spindle speed to the specified spindle speed exceeds the spindle speed fluctuation of R, the actual spindle speed is regarded as being so fast that an overheat can occur.

I: Spindle speed fluctuation width at which the actual spindle speed (min^{-1}) is so fast that an overheat can occur
If the fluctuation (width) between the specified and actual spindle speeds exceeds the spindle speed fluctuation width of I, the actual spindle speed is regarded as being so fast that an overheat can occur.

G26 enables the spindle speed fluctuation detection function. The values specified for P, Q, R, and **I** are set in the following parameters: No. 4914, No. 4911, No. 4912, and No. 4913, respectively. Each command address corresponds to a parameter number as listed below.

Command address	Parameter number
Q	No.4911
R	No.4912
I	No.4913
P	No.4914

If the P, Q, R, or I command address is omitted, the function detects the fluctuation of the actual spindle speed according to the value set in the corresponding parameter (No. 4914, No. 4911, No. 4912, or No. 4913).
The parameters (No. 4914, No. 4911, No. 4912, and No. 4913) for the spindle on which the currently selected position coder is mounted are used for the setting and spindle speed fluctuation detection check.

- Spindle fluctuation detection off

G25;

G25 disables the spindle speed fluctuation detection function.
When G25 is specified, the parameters (No. 4914, No. 4911, No. 4912, and No. 4913) are unchanged. When the power is turned on or after a reset (clear state (bit 6 (CLR) of parameter No. 3402 = 1)) is executed, the spindle speed fluctuation detection function is disabled (G25). For the clear state, also check the setting of bit 3 (C19) of parameter No. 3408 for the M series or bit 0 (C08) of parameter No. 3407 for the T series.

Explanation

The function for detecting spindle speed fluctuation checks whether the actual speed varies for the specified speed or not. Si or Sr, whichever is greater, is taken as the allowable fluctuation speed (Sm). An alarm (OH0704) is activated when the actual spindle speed varies for the commanded speed (Sc) under the condition that the variation width exceeds the allowable variation width (Sm).

$|Sc - Sa| > Sm$

Sc : Specified spindle speed
Sa : Actual spindle speed
Si : The allowable constant variation width which is independent of the specified spindle speed (parameter (No.4913))
Sr : The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio). (r = parameter (No.4912))

Parameter FLR(No.4900#0)= 0	Parameter FLR(No.4900#0)= 1
$Sr = Sc \times \frac{r}{100}$	$Sr = Sc \times \frac{r}{1000}$

Sm: Si or Sr, whichever is greater

- Conditions to start spindle speed fluctuation detection

If the specified spindle speed Sc changes, spindle speed fluctuation detection starts when one of the conditions below is met:

<1> The actual spindle speed falls in a range of (Sc - Sq) to (Sc + Sq)
Sc : Specified spindle speed
Sq : Tolerance within which the spindle is assumed to attain the programmed speed
(parameter (No.4911))

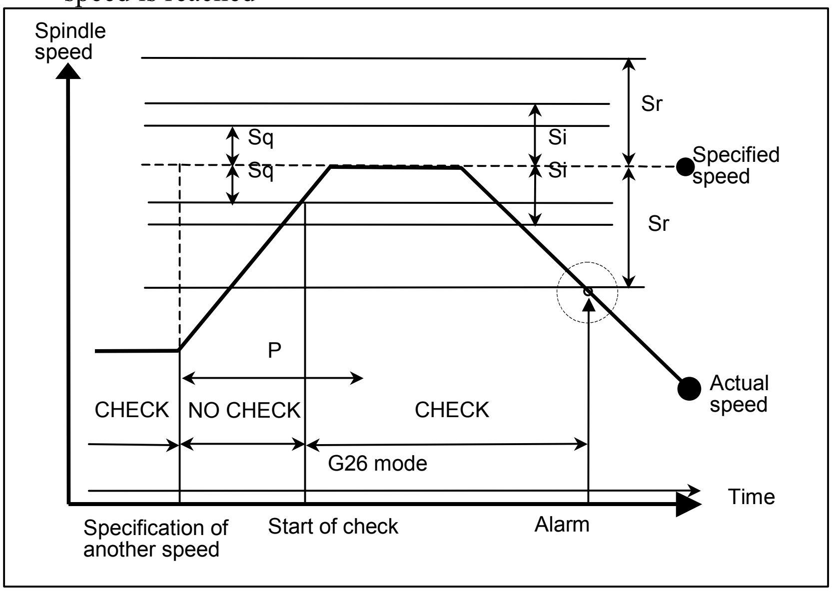
Parameter FLR= 0	Parameter FLR= 1
$Sq = Sc \times \frac{q}{100}$	$Sq = Sc \times \frac{q}{1000}$

<2> When time p specified in parameter No. 4914 elapses after the specified speed Sc changes.

- Examples of spindle speed fluctuation detection

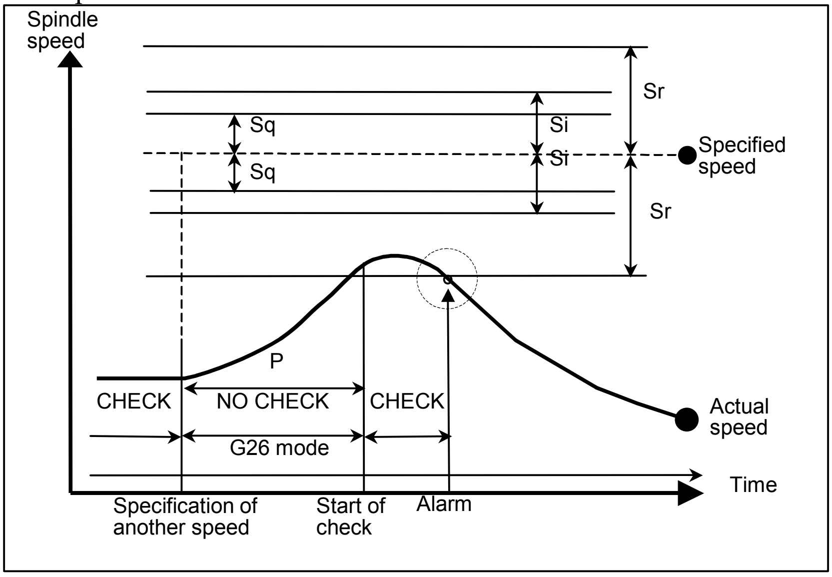
(Example 1)

When an alarm (OH0704) is issued after a specified spindle speed is reached



(Example 2)

When an alarm (OH0704) is issued before a specified spindle speed is reached



Specified speed :

(Speed specified by address S and five-digit value) × (spindle override)

Actual speed : Speed detected with a position coder

p : Period after a change occurs in the actual spindle speed until detection starts

Parameter No.4914, address P

Sq : (Specified spindle speed) × (Detection start tolerance (q))

Parameter No.4911, address Q

Parameter FLR = 0	Parameter FLR = 1
q ------- 100	q ------- 1000

Sr : (Specified spindle speed) × (Allowable variation (r))

Parameter No.4912, address R

Parameter FLR = 0	Parameter FLR = 1
r --------- 100	r ----------- 1000

Si : Allowable variation width

Parameter No.4913, address I

If the difference between the specified speed and actual speed exceeds both Sr and Si, an alarm (OH0704) is raised.

- Relationship between spindle speed control and each spindle

Spindle / Function	Serial spindle			
	1st spindle	2nd spindle	3rd spindle	4th spindle
Spindle speed fluctuation detection	Possible	Possible(*1)	Possible(*1)	Possible(*1)

NOTE

1 An optional function of multi spindle control is necessary.

2 The spindle speed fluctuation detection function is effective for a single spindle. The function cannot be executed for two or more spindles.
The spindle speed fluctuation detection function is effective for a spindle on which the currently selected position coder is mounted. Just a single position coder can be selected. Multiple position coders cannot be selected. For the selection of a position coder, see the section of "Multi spindle."

* Position coder selection signals (PC2SLC<Gn028.7>, PC3SLC<Gn026.0>, PC4SLC<Gn026.1>)

3 The parameters that become valid are the parameters of the spindle speed fluctuation detection function (No.4911, No.4912, No.4913, No.4914) for the spindle on which the currently selected position coder is mounted.

- Spindle for which to detect the spindle speed fluctuation

For the spindle for which to detect the spindle speed fluctuation, refer to the appropriate manual provided by the machine tool builder.

10 TOOL FUNCTION (T FUNCTION)

10.1 TOOL SELECTION FUNCTION

By specifying an up to 8-digit numerical value following address T, a code signal and a strobe signal are transmitted to the machine tool. This is used to select tools on the machine.
One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commendable with address T and the correspondence between the T codes and machine operations.
When a move command and a T code are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and T function commands.

(ii) Executing T function commands upon completion of move command execution.

The selection of either (i) or (ii) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

Explanations

T

The value after the T code indicates the desired tool. Part of the value is also used as a tool offset number that specifies the amount of tool offset or the like. The tool can be selected as follows according to the specification method and parameter settings.

<table>
<tr><th colspan="2">Description of a T code (Note 1)</th><th rowspan="2">How to specify the offset number for each parameter setting (Note 2)</th></tr>
<tr><th>LGN (No.5002#1) = 0</th><th>LGN (No.5002#1) = 1</th></tr>
<tr><td rowspan="2">T○○○○○○○ ○
↑ ↑
Tool selection: ○○○○○○○
Tool geometry tool wear offset: ○</td><td rowspan="2">T○○○○○○○ ○
↑ ↑
Tool selection tool geometry offset: ○○○○○○○
Tool wear offset: ○</td><td>The tool wear offset number is specified using the low-order one digit of a T code.</td></tr>
<tr><td>When parameter (No.5028) is set to 1</td></tr>
<tr><td rowspan="2">T○○○○○○ ○○
↑ ↑
Tool selection: ○○○○○○
Tool geometry tool wear offset: ○○</td><td rowspan="2">T○○○○○○ ○○
↑ ↑
Tool selection tool geometry offset: ○○○○○○
Tool wear offset: ○○</td><td>The tool wear offset number is specified using the low-order two digits of a T code.</td></tr>
<tr><td>When parameter (No.5028) is set to 2</td></tr>
<tr><td rowspan="2">T○○○○○ ○○○
↑ ↑
Tool selection: ○○○○○
Tool geometry tool wear offset: ○○○</td><td rowspan="2">T○○○○○ ○○○
↑ ↑
Tool selection tool geometry offset: ○○○○○
Tool wear offset: ○○○</td><td>The tool wear offset number is specified using the low-order three digits of a T code.</td></tr>
<tr><td>When parameter (No.5028) is set to 3</td></tr>
</table>

NOTE

1. The maximum number of digits of a T code can be specified by parameter (No.3032) as 1 to 8.
2. When parameter (No.5028) is set to 0, the number of digits used to specify the offset number in a T code depends on the number of tool offsets.
   Example) When the number of tool offsets is 1 to 9: Low-order one digit
   When the number of tool offsets is 10 to 99: Low-order two digits
   When the number of tool offsets is 100 to 999: Low-order three digits

Refer to the machine tool builder's manual for correspondence between the T-code and the tool and the number of digit to specify tool selection.

Example (T2+2)

```
N1G00X1000Z1400;
N2T0313;   (Select tool No. 3 and tool offset value No. 13)
N3X400Z1050;
```

10.2 TOOL MANAGEMENT FUNCTION

Overview

The tool management function totally manages tool information including information about tool offset and tool life.

Explanation

A tool type number is specified with a T code. The tool type number is any number the user can define freely. With tool type numbers, tools can be grouped by various conditions such as life, compensation value, and cutting conditions. When each type is assumed to have a single tool, tool type numbers are equivalent to unique tool numbers.
For each tool, an information storage area is prepared in the CNC (SRAM). This area contains information such as a tool type number, tool life, tool status (such as a breakage condition), tool compensation number (H, D, G, or W), spindle speed (S), cutting feedrate (F), and freely-definable customize data. Such data is called tool management data.
A cartridge management table that links cartridge information and tool management data is provided so that the CNC can manage the cartridges of the machine and tool change operations. In addition, areas for managing the tools in the spindle and tool standby positions are prepared.
When a tool type number is specified using a T code command, a tool that has the tool type number and the shortest life is searched for, and the cartridge number and pot number of the location where the tool is stored are output to the PMC. Then, a tool change operation using the cartridge number and pot number is enabled by the PMC ladder sequence.
Tool information in the CNC is managed by the tool management data and cartridge management table (including the spindle management table and standby position table).

- Tool management data

As tool management data, information about each tool is registered with a tool management data number.
The following number of sets of tool management data can be used:

Tool management function 64 sets	64 sets in total
Tool management function 240 sets	240 sets in total
Tool management function 1000 sets	1000 sets in total

NOTE
For the number of tool management data sets, refer to the relevant manual issued by the machine tool builder.

- Details of data

The following details the tool management data registered for each data number:

- Tool type number (T code)

Item	Description
Data length	4byte
Valid data range	0,1 to 99,999,999

- Tool life counter

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

The value of an increment counter, that is, the number of use times (time) is indicated. The remaining life value is [the maximum tool life value minus tool life counter value].

- Maximum tool life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

- Notice life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

If a value other than 0 is set, the tool life expiration notice signal TLCHB <F064#3> or TLCHBx <F329#4 to #7> is output when the remaining life value of the tool (= maximum tool life value minus tool life counter value) has reached the set value.

- Tool life status

Item	Description
Data length	1byte
Detail data	0: Life management is not performed. 1: Tool not yet used 2: Life remains. 3: Life expired. 4: Tool breakage (skip)

The machine (PMC) determines tool breakage and stores corresponding information through the window. In tool management of the CNC, a broken tool is regarded as being equivalent to tools whose lives have expired.

- Tool information

Item	Description
Data length	1 byte (flag data)
#0 RGS	0: Tool management data is invalid. (-) 1: Tool management data is valid. (R)
#1 TIM	0: Type for counting the number of use times (C) 1: Type for counting time (T)
#2 BDT	0: Normal tool (N) 1: Tool with a large diameter (B)
#3 LOC	0: Data accessible (U) 1: Data not accessible (L)
#4 SEN	When the tool life status indicates that tool management is not performed: 0: This tool is not searched. (-). 1: This tool is searched. (S)
#5 to #7	Reserved

When RGS is set to 0 in tool management data, the tool management data is regarded as not being registered even when values are set for other items.

NOTE

Make sure you specify the same life count type for all tools of the same type. You can check tools of the same type for any difference in life count type, using the "check function."

M

- Tool length compensation number (H)

Item	Description
Data length	2byte
Valid data range	0 to 999

- Cutter compensation number (D)

Item	Description
Data length	2byte
Valid data range	0 to 999

T

- Tool geometry compensation number (G)

Item	Description
Data length	2byte
Valid data range	0 to 999

- Tool wear compensation number (W)

Item	Description
Data length	2byte
Valid data range	0 to 999

NOTE
When the machine control type is the combined system type, tool length compensation and cutter compensation numbers are used for paths for the machining center system, and for paths for the lathe system, tool geometry compensation and tool wear compensation numbers are used.

- Spindle speed (S)

Item	Description
Data length	4byte
Unit of data	min^{-1}
Valid data range	1 to 99,999

- Feedrate (F)

Item	Description
Data length	4byte
Unit of data	mm/min, inch/min, deg/min, mm/rev, inch/rev
Valid data range	0 to 99,999,999

As additional tool management data, areas for setting customize data (5 data items including customize data 0 and customize data 1 to 4) are provided without defining specific usage. The user can use these customize data areas freely to set a warning life value, cutting resistance, override value, spindle current value, maximum and minimum S/F, and other items according to the targeted application.

- Customize data 0

Item	Description
Data length	1 byte (bit type)
Valid data range	0 or 1 on a bit-by-bit basis

- Customize data 1 to 4 (to 20) (to 40)

Item	Description
Data length	4byte
Valid data range	-99,999,999 to 99,999,999

> **NOTE**
> For the maximum number of tool management function customization data items, refer to the relevant manual issued by the machine tool builder.

- Cartridge management table

The storage status of tools in cartridges is managed with the cartridge management table.

- Multiple cartridge numbers can be defined. (Numbers from 1 to 4, up to four cartridges)
- The maximum number of pots for all cartridges is 64, 240, or 1000, which depends on whether a tool management data option is selected.
- Cartridge numbers and pot numbers can be assigned freely within cartridge management data by parameter setting (described later).
- The tool management data number adjacent to a pot number is linked with tool data defined in the tool management table. Therefore, the tool attached to the pot is indicated.
- Zero set as a data number indicates that no tool is attached.
- The cartridge management table can be read from and written to through the PMC window and FOCAS2.
- The spindle management table and standby position table are provided to indicate special cartridge positions.
- Spindle positions and standby positions, regarded as special cartridge positions, have fixed cartridge numbers 11 to 14 (the positions of the first to fourth spindles) and 21 to 24 (the first to fourth standby positions).
- With the PMC window, the spindle position table and standby position table can be read from and written to.
- Tool life counting is performed only for the tools at the spindle positions.

- Multi-path system

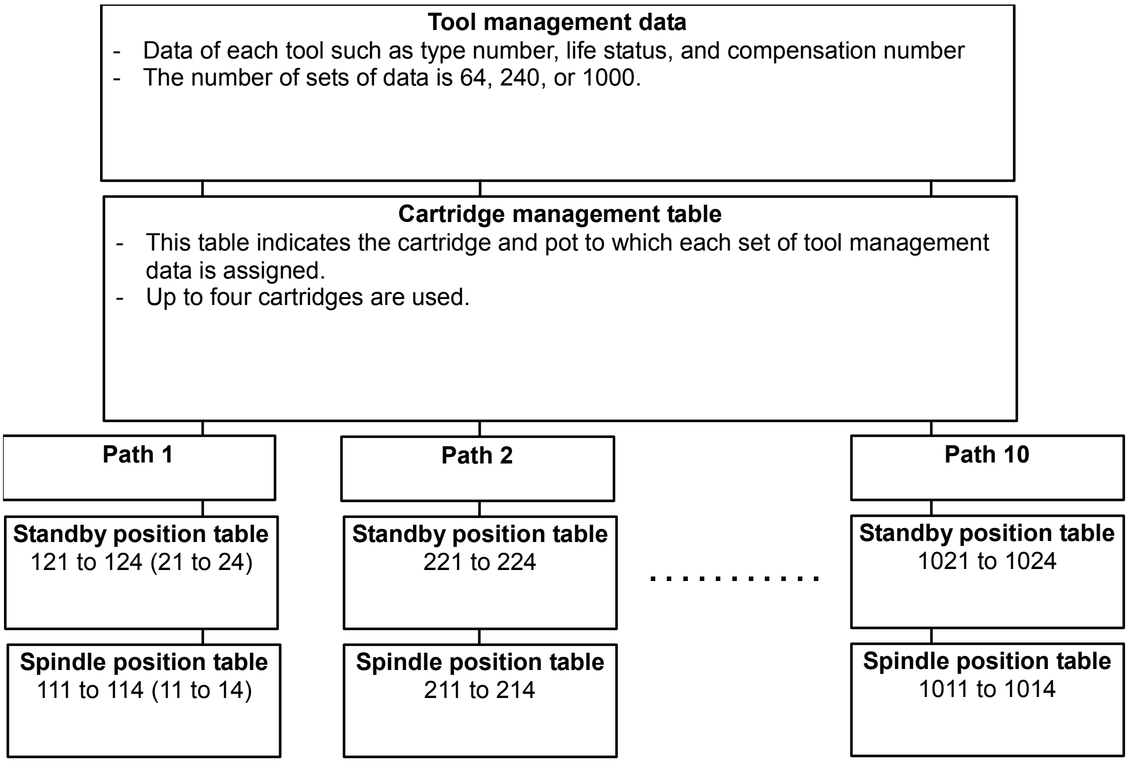
The tool management data and cartridge management table are common data among the paths. The spindle management table and standby position table, however, are treated as independent data for each path.
When the spindle table or standby position table is specified as a cartridge from the PMC window, specify the following, in which the path number is set in the hundred's place:

	Spindle position			
	First	**Second**	**Third**	**Fourth**
First path	111(11)	112(12)	113(13)	114(14)
Second path	211	212	213	214
Third path	311	312	313	314
Fourth path	411	412	413	414
Fifth path	511	512	513	514
Sixth path	611	612	613	614
Seventh path	711	712	713	714
Eighth path	811	812	813	814
Ninth path	911	912	913	914
Tenth path	1011	1012	1013	1014

	Standby position			
	First	**Second**	**Third**	**Fourth**
First path	121(21)	122(22)	123(23)	124(24)
Second path	221	222	223	224
Third path	321	322	323	324
Fourth path	421	422	423	424
Fifth path	521	522	523	524
Sixth path	621	622	623	624
Seventh path	721	722	723	724
Eighth path	821	822	823	824
Ninth path	921	922	923	924
Tenth path	1021	1022	1023	1024

NOTE

When specifying 111, 121, and so on to specify the first path, you may specify just 11, 21, and so on.



- M/T series

The tool management data and cartridge management table are data common to the M and T series.

- Tool life management and tool change

The CNC performs tool life management, regarding tools having the same tool type number as a group. When a tool type number (T code) is specified by an NC program, the tool management data registered in the CNC is searched to find a tool having the shortest life among the tools with the same tool type number.
The cartridge number and pot number corresponding to the searched tool are output as a T code signal to the PMC. Based on the output cartridge number and pot number, the PMC performs preparation for a tool change (to the next tool). A different tool can also be selected on the PMC side.
The CNC performs tool life counting for each tool that is at a spindle position in the spindle management table.

When the lives of all tools having the tool type number specified by the T code have expired, alarm PS5317 is issued. If there is a tool in the spindle position or standby position, that tool is selected to continue machining.
As the T code signal, a specified tool type number instead of the cartridge number and pot number can also be output directly by parameter setting.

There are two types of tool life management counting methods: counting the number of use times and counting cutting time. One of the counting methods is set in tool information of tool management data.

Other major specifications related to tool life management are as follows:
Tool type number (T code): Up to 8 digits (1 to 99,999,999)
Maximum tool life value:
    99,999,999 times when the number of use times is specified
    999 hours 59 minutes 59 seconds when time is specified
Life count interval when time is specified: 1 second
Tool life management count restart M code: Enabled
Tool life count override: Enabled

To specify a certain tool directly without performing tool life management by tool type number, use the following format:

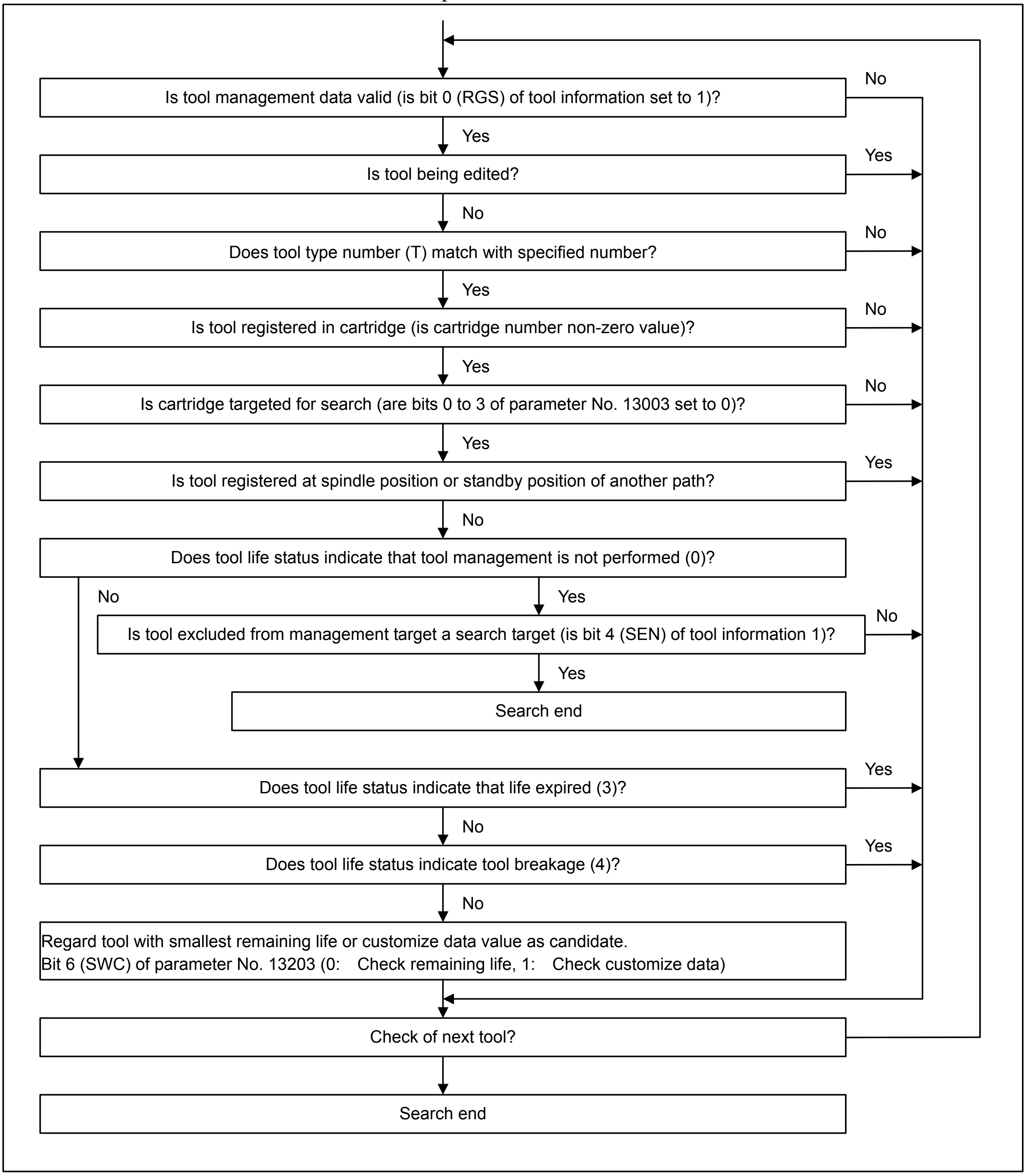
M (value in parameter No. 13252) T (cartridge number) (pot number) ;

When parameter No. 13252 is set to 333, the cartridge number is 2, and the pot number is 27, for example, the following command is specified:

M333 T20027;

- Tool search order

Tools having a tool type number (T) specified by a program are searched sequentially from tool management data number 1 while registered data contents are checked. The following shows how a search operation is made within the NC:



- System variables

The following tool management data of the tool being used as a spindle after a tool change by M06 and the tool to be used next which is specified by a T code can be read through custom macro variables:

Being used	Item	
#8401	Tool management data number	
#8402	Tool type number (T)	
#8403	Tool life counter	
#8404	Maximum tool life value	
#8405	Tool notice life value	
#8406	Tool life status	
#8407	Customize data 0 (bit)	
#8408	Tool information	
#8409	Tool length compensation number (H)	For machining center systems
#8410	Cutter compensation number (D)	For machining center systems
#8411	Spindle speed (S)	
#8412	Cutting feedrate (F)	
#8413	Tool geometry compensation number (G)	For lathe systems
#8414	Tool wear compensation number (W)	For lathe systems
#8431	Customize data 1	
#8432	Customize data 2	
#8433	Customize data 3	
#8434	Customize data 4	
#8435	Customize data 5	
#8436	Customize data 6	
#8437	Customize data 7	
#8438	Customize data 8	
#8439	Customize data 9	
#8440	Customize data 10	
#8441	Customize data 11	
#8442	Customize data 12	
#8443	Customize data 13	
#8444	Customize data 14	
#8445	Customize data 15	
#8446	Customize data 16	
#8447	Customize data 17	
#8448	Customize data 18	
#8449	Customize data 19	
#8450	Customize data 20	
#8451	Customize data 21	
#8452	Customize data 22	
#8453	Customize data 23	
#8454	Customize data 24	
#8455	Customize data 25	
#8456	Customize data 26	
#8457	Customize data 27	
#8458	Customize data 28	
#8459	Customize data 29	
#8460	Customize data 30	
#8461	Customize data 31	
#8462	Customize data 32	
#8463	Customize data 33	

Being used	Item
#8464	Customize data 34
#8465	Customize data 35
#8466	Customize data 36
#8467	Customize data 37
#8468	Customize data 38
#8469	Customize data 39
#8470	Customize data 40

When a cartridge number of a spindle position (11 to 14) or standby position (21 to 24) is specified in #8400, information about the corresponding position can be read.
If the spindle position table or standby position table has an empty pot, <empty> is read from #8402 to #8470.
Value 0 is read from #8401 (data number).

Therefore, machining conditions registered in tool management data can be specified directly by coding, for example, D#8410, H#8409, S#8411, and F#8412 with a tool change macro (such as M06). Similarly, customize data can be referenced by a custom macro, and machining programs can be customized according to the tool used.

NOTE
For the maximum number of tool management function customization data items, refer to the relevant manual issued by the machine tool builder.

- Specifying a tool compensation number

M

When parameter No. 13265 is 0, a compensation number registered as tool management data of a tool attached at a spindle position can be selected by specifying H99 or D99.
(99 is treated as a special number, so 99 cannot be specified directly as a compensation number.)
When other than 0 is set in parameter No. 13265, the number set in the parameter can be specified instead of 99. For example, if parameter No. 13265 is 3, specifying H3 specifies the tool length compensation number registered for the tool at the spindle position.

T

When the tool management function is not used, a tool compensation number is specified also with a T code; when the tool management function is used, the T code is used only to specify a tool type number, so a tool compensation number must be specified with address D.
Except the specifications for addresses, the specifications for the number of permissible digits (parameter No. 3032) and the number of digits consisting of a compensation number (parameter No. 5028), separation of geometry compensation numbers and wear compensation numbers (bit 1 (LGN) of parameter No. 5002), and so on are the same as for T.

If parameter No. 13265 is set to 0, when a compensation number registered for a tool attached at a spindle position is specified, the specification format varies according to the number of digits as follows, which is the same as for conventional T:
When the compensation number is 1 digit long: D9
When the compensation number is 2 digits long: D99
When the compensation number is 3 digits long: D999
Because 9, 99, or 999 is treated as a special number, it cannot be specified directly as a compensation number.
Unless parameter No. 13265 is set to 0, the number set in the parameter can be specified instead of 9, 99, or 999. When parameter No. 13265 is set to 3, specifying D3 specifies the tool geometry compensation number and tool wear compensation number registered for a tool attached at the spindle position.

Multi-path system

Depending on whether the local path is a machining center system or a lathe system, tool compensation numbers are specified by using one of the above methods.

Spindle selection

When specifying compensation numbers of a tool attached to a spindle other than the first spindle, specify the spindle number with address P within the same block that contains H/D. When specifying the first spindle, you can omit P.
D99 P3; Specifying compensation numbers registered for the tool attached at the third spindle
D99 ; Specifying compensation numbers registered for the tool attached at the first spindle

- Read/write operations for tool management data and cartridge management table

The tool management data and cartridge management table can be read from and written to by using the CNC/PMC data window library (FOCAS2). Therefore, a specific tool management system including all available tool data not registered in the CNC can be built easily by using the OPEN CNC.

Similarly, the tool management data and cartridge management table can also be read from and written to using the PMC window.

Tool management data of the tool being used as a spindle after a tool change operation performed by M06 and the tool being selected by specifying a T code can be read using a custom macro.

The user can modify tool management data by MDI from the tool management function screen of the CNC. Addition, modification, and deletion of the above tool management data can be made from a part program (G10). Such data can also be input to and output from external I/O equipment by using the ALL I/O screen and tool management function screen.

- G10 format

Addition, modification, and deletion are performed for the tool management data and cartridge management table from programs. For use of this function, the programmable data input option function is required.
If a format error is found in the commands from G10 L75/L76/L77 to G11, or if a value beyond the valid data range is specified, alarm PS5312 is issued. In such a case, correct the program. Within the range from G10 to G11, no decimal point can be specified with any address. If a decimal point is specified, alarm PS0007 results.

- Registering new tool management data

Tool management data can be registered. When data is punched out to an external device from the tool management data screen, this format is used.
The specification of those items that are not registered may be omitted.

```
G10 L75 P1;
N_ ; Tool management data number specification
T_ C_ L_ I_ B_ Q_ H_ D_ S_ F_ J_ K_ ;
P0 R_ ;   Customization data 0
P1 R_ ;   Customization data 1
P2 R_ ;   Customization data 2
P3 R_ ;   Customization data 3
P4 R_ ;   Customization data 4
N_ ;      Tool management data number
 :
G11;
    N_   Tool management data No. 1 to 64
         (1 to 240, 1 to 1000)
    T_   Tool type No. (T)       0 to 99,999,999
    C_   Tool life counter       0 to 99,999,999
    L_   Maximum tool life       0 to 99,999,999
    I_   Noticed life            0 to 99,999,999
    B_   Tool life state         0 to 4
    Q_   Tool information        Bit format (8 bits)
    H_   Tool length compensation No. (H)
         0 to 999 (M series)
    D_   Cutter compensation No. (D)
         0 to 999 (M series)
    S_   Spindle speed (S)       0 to 99,999
    F_   Cutting feedrate (F)    0 to 99,999,999
    J_   Tool geometry compensation No. (G)
         0 to 999 (T series)
    K_   Tool geometry compensation No. (W)
         0 to 999 (T series)
    P_   Customization data No. 0 to 4 (0 to 20, 0 to 40)
    R_   Customization data value
         -99,999,999 to 99,999,999
```

Specify customization data in the following format:

P (customization-number) R (value)

Use the bit format only when specifying the customization data 0 (P0). Specify other data in the binary format. The specification of customization data that need not be set may be omitted.

Example)
G10 L75 P1;
N1 ; Tool management data No.1
T10000001 C0 L1000 B1 Q00000001 H1 D1 S4000 F10000 ;
P0 R11101101 ; Customization data 0
P4 R99999999 ; Customization data 4
N2 ; Tool management data No.2
:
G11 ;

Modifying tool management data

Tool management data can be modified.
The specification of those items that are not modified may be omitted.

```
G10 L75 P2 ;
N_ ;
T_ C_ L_ I_ B_ Q_ H_ D_ S_ F_ J_ K_ ;
P_ R_ ;
N_ ;
 :
G11 ;
```

Deleting tool management data

The data of a specified data number can be deleted from tool management data.
The cartridge management table data corresponding to a deleted tool management data number is also deleted. (The tool management data number in the cartridge management table is cleared to 0.)

```
G10 L75 P3 ;
N_ ;
N_ ;
 :
N_ ;
G11 ;
```

Registering new cartridge management table data

A tool management data number can be registered with a free pot in the cartridge management table.

G10 L76 P1 ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
G11 ;

For a spindle position table and standby position table, only cartridge number data is specified.

Example)

G10 L76 P2 ;

N11 R1; Changes the tool management data number of the spindle position to No. 1.

N21 R29; Changes the tool management data number of the standby position to No. 29.

G11 ;

Modifying the cartridge management table

Tool management data numbers in the cartridge management table can be modified.

G10 L76 P2 ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
G11 ;

For a spindle position table and standby position table, only cartridge number data is specified.

Example)

G10 L76 P2 ;

N11 R1; Changes the tool management data number of the spindle position to No. 1.

N21 R29; Changes the tool management data number of the standby position to No. 29.

G11 ;

Deleting cartridge management table data

Tool management data numbers can be deleted from the cartridge management table.

```
G10 L76 P3 ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
G11 ;
```

For a spindle position table and standby position table, only cartridge number data is specified.

Example)

G10 L76 P3 ;

N11 ; Deletes the tool management data number of the spindle position (clears the number to 0).

N21 ; Deletes the tool management data number of the standby position (clears the number to 0).

G11 ;

Naming customization data

The display name of customization data (0 to 40) can be set.

```
G10 L77 P1 ;
N_ ;
P_ R_ ;
P_ R_ ;
;
N_ ;
P_ R_ ;
P_ R_ ;
G11 ;

   N_:  Customization data No. (0 to 40)
   P_:  Character No. (1 to 16)
   R_:  Character code (ANK or shifted JIS)
        - When a shifted JIS code is used, an area for
          two characters is used.
        - Specify 0 to clear data.
```

- A set display name can be checked on the tool management data screen only.
- When no name is registered, a name such as the default "Customize 3" is displayed.
- A name consisting of up to 16 characters is displayed. For a name shorter than 16 characters, register 0 in the empty area. Those characters that are registered immediately before 0 are displayed.

- To clear data, set 0 as a character code.
- No character code data check is made.

When the name of customization data 3 is set as "測定値５", for example, specify the following:

Example)

```
G10 L77 P1 ;
N3 ;              Specifies customization data 3.
P1 R37290 ;   Shifted JIS code 91AAh for "測"
P3 R37608 ;   Shifted JIS code 92E8h for "定"
P5 R37484 ;   Shifted JIS code 926Ch for "値"
P7 R33364 ;   Shifted JIS code 8254h for "５"
P9 R0 ;          Clears data.   (Not displayed.   End)
G11 ;
```

Naming tool life states

The display name of a tool life state (0 to 4) can be set.

```
G10 L77 P2 ;
N_ ;
P_ R_ ;
P_ R_ ;
N_ ;
P_ R_ ;
P_ R_ ;
G11 ;

   N_:  Tool life state (0 to 4)
   P_:  Character No. (1 to 12)
   R_:  Character code (ANK or shifted JIS)
        - When a shifted JIS code is used, an area for
          two characters is used.
        - Specify 0 to clear data.
```

- A set display name can be checked on the tool management data screen only.
- If no name is registered, a default such as "Remaining" and "Unremaining" is displayed.
- A name consisting of up to 12 characters is displayed. For a name shorter than 12 characters, register 0 in the empty area. Those characters that are registered immediately before 0 are displayed.
- To clear data, set 0 as a character code.
- No character code data check is made.
- The defaults are as follows:
  - 0: Invalid
  - 1: Remaining
  - 2: Remaining
  - 3: Unremaining
  - 4: Broken

To set "正常" as the name of tool life state 2 (Remaining), specify the following:

Example)

G10 L77 P2 ;
N2 ; Specifies tool life state 2.
P1 R37043 ; Shifted JIS code 90B3h for "正"
P3 R36845 ; Shifted JIS code 8FEDh for "常"
P5 R0 ; Clears data. (Not displayed. End)
G11 ;

10.3 TOOL MANAGEMENT EXTENSION FUNCTION

Overview

The following functions have been added to the tool management function:

1. Customization of tool management data display
2. Setting of spindle position/standby position display
3. Input of customize data with the decimal point
4. Protection of various tool information items with the KEY signal
5. Selection of a tool life count period
6. Individual data screen
7. Total life time display for tools of the same type

They will be explained in detail below.

10.3.1 Customization of Tool Management Data Display

With the tool management data screen display customization function, the display positions of screen elements (type number, tool information, life counter, and so forth) on the tool management screen can be changed and whether to display or hide such screen elements can be chosen using the G10 format. This function enables a customized tool management screen to be configured.

Format

```
G10L77P3;
N_ R_;
G11;
  N_: Tool management data screen display position number
  R_: Tool management data display item
```

Explanation

- number (N_)

- A display position number represents an ordinal number from the leftmost position on the tool management data screen.
- As N, a number from 1 to 200 can be specified.

- Tool management data display item (R_)

- For a value to be set (in R_), see the table below.
- For actual setting, consider a display width plus 1.

Items common to machining center systems and lathe systems

R	Item	Display width	Remarks
-1	End of setting	-	
0	Blank column	10	
1	No.	4	For spindle/standby positions for 10 paths
2	TYPE-NO.	8	
3	MG	4	
4	POT	5	
5	T-INFORMATION	10	

R	Item	Display width	Remarks
6	L-COUNT	10	
7	MAX-LIFE	10	
8	NOTICE-L	10	
9	L-STATE	6 or 12	The display width is switched by bit 1 of parameter No. 13201.
10	S (Spindle speed)	10	
11	F (Feedrate)	10	
12	Tool figure number (A)	3	

Offset-related items for machining center systems

R	Item	Display width	Remarks
20	H (Tool length compensation number)	4	
21	D (Cutter compensation number)	4	
22	OFFSET－M	10	Tool offset memory A
23	GEOMETRY－M	10	Tool offset memory B
24	WEAR－M	10	
25	GEOMETRY（H）	10	Tool offset memory C
26	WEAR（H）	10	
27	GEOMETRY（D）	10	
28	WEAR（D）	10	

Offset-related items for lathe systems

R	Item	Display width	Remarks
40	TG (Tool geometry compensation number)	4	
41	TW (Tool wear compensation number)	4	
42	OFFSET－X	10	
43	OFFSET－Z	10	
44	OFFSET－R	10	Tool nose radius compensation
45	TIP	10	
46	OFFSET－Y	10	Y-axis offset
47	OFFSET－B	10	B-axis control
48	GEOMETRY (X)	10	Tool geometry/wear compensation
49	WEAR (X)	10	
50	GEOMETRY (Z)	10	
51	WEAR (Z)	10	
52	GEOMETRY (R)	10	Tool nose radius compensation, Tool geometry/wear compensation
53	GEOMETRY TIP	10	
54	WEAR (R)	10	
55	WEAR TIP	10	
56	GEOMETRY (Y)	10	Y-axis offset, Tool geometry/ wear compensation
57	WEAR (Y)	10	
58	GEOMETRY (B)	10	B-axis control, Tool geometry/ wear compensation
59	WEAR (B)	10	
60	GEOMETRY (X2)	10	2nd geometry tool offset
61	GEOMETRY (Z2)	10	
62	GEOMETRY (Y2)	10	

Items related to customize data

R	Item	Display width	Remarks
80	CUSTOM 0	10	
81	CUSTOM 1	10	
82	CUSTOM 2	10	
83	CUSTOM 3	10	
84	CUSTOM 4	10	
85	CUSTOM 5	10	Tool management function customize data extension (5 to 20) or tool management function customize data extension (5 to 40)
86	CUSTOM 6	10	
87	CUSTOM 7	10	
88	CUSTOM 8	10	
89	CUSTOM 9	10	
90	CUSTOM 10	10	
91	CUSTOM 11	10	
92	CUSTOM 12	10	
93	CUSTOM 13	10	
94	CUSTOM 14	10	
95	CUSTOM 15	10	
96	CUSTOM 16	10	
97	CUSTOM 17	10	
98	CUSTOM 18	10	
99	CUSTOM 19	10	
100	CUSTOM 20	10	
101	CUSTOM 21	10	Tool management function customize data extension (5 to 40)
102	CUSTOM 22	10	
103	CUSTOM 23	10	
104	CUSTOM 24	10	
105	CUSTOM 25	10	
106	CUSTOM 26	10	
107	CUSTOM 27	10	
108	CUSTOM 28	10	
109	CUSTOM 29	10	
110	CUSTOM 30	10	
111	CUSTOM 31	10	
112	CUSTOM 32	10	
113	CUSTOM 33	10	
114	CUSTOM 34	10	
115	CUSTOM 35	10	
116	CUSTOM 36	10	Tool management function customize data extension (5 to 40)
117	CUSTOM 37	10	
118	CUSTOM 38	10	
119	CUSTOM 39	10	
120	CUSTOM 40	10	

NOTE

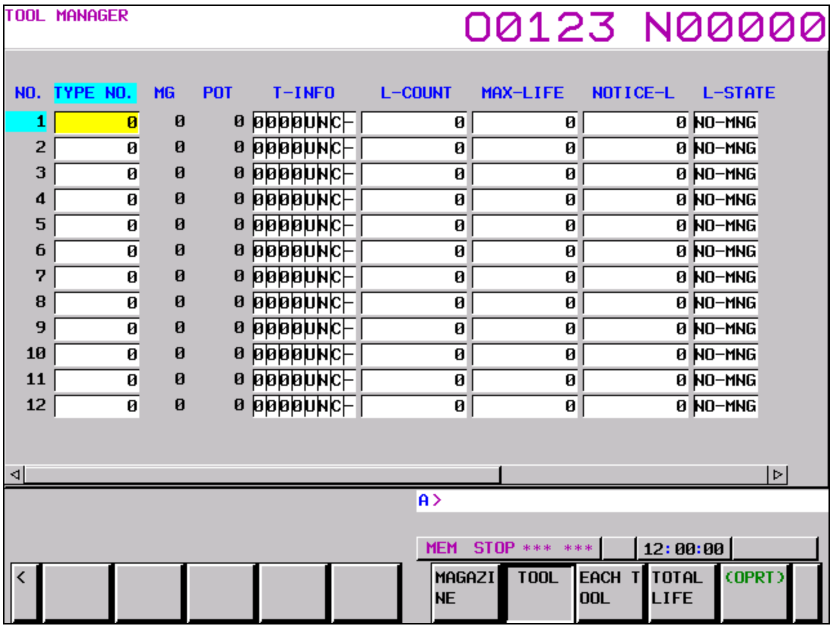
1. If G10 L77 P3 is terminated normally, the power must be turned off before operation is continued.
2. The setting becomes effective after the power is turned off then back on.

Example

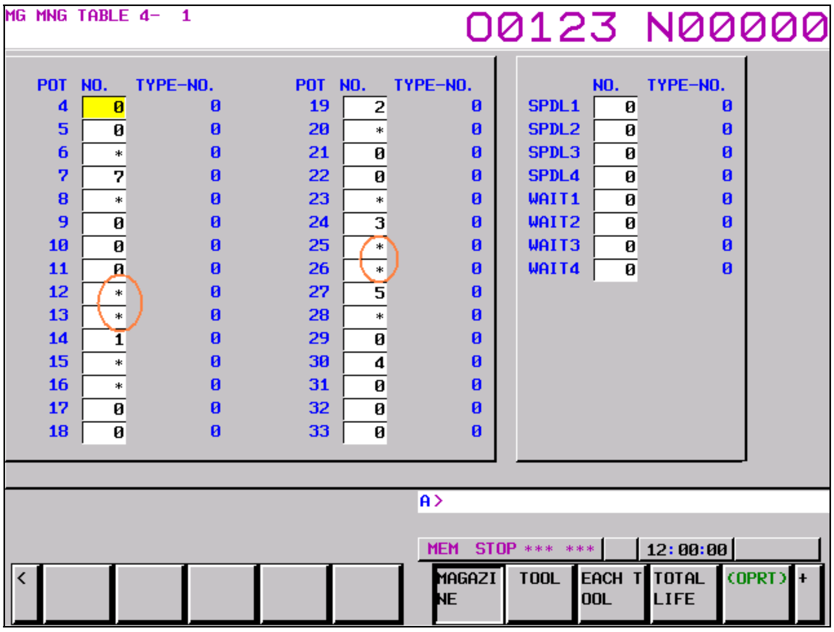
Example of setting tool offset memory A

G10L77P3;	Set tool management data screen display customization
N1 R1;	Set No. as number 1
N2 R2;	Set TYPE-NO. as number 2
N3 R3;	Set MG as number 3
N4 R4;	Set POT as number 4
N5 R5;	Set T-INFORMATION as number 5
N6 R6;	Set L-COUNT as number 6
N7 R7;	Set MAX-LIFE as number 7
N8 R8;	Set NOTICE-L as number 8
N9 R9;	Set L-STATE as number 9
N10 R11;	Set F (Feedrate) as number 10
N11 R21;	Set D as number 11
N12 R22;	Set OFFSET-M as number 12
N13 R80;	Set CUSTOM 0 as number 13
N14 R81;	Set CUSTOM 1 as number 14
N15 R-1;	End
G11;	Cancel the setting mode

On the tool management data screen, the first page displays "No., Type Number, MG, Pot, Tool Information, Life Counter, Maximum Life, Noticed Life, and Life State".
The second page displays "F (Feedrate), D (Cutter Compensation Number), Tool Offset-M, Customize Data 0, and Customize Data 1".



Example 1: Page 1



Example 1: Page 2

NOTE

1. This setting is enabled when bit 0 (TDC) of parameter No. 13201 is set to 1.
2. Up to 20 pages can be set.
3. Be sure to specify an end.
4. If an item that requires the corresponding option is specified without specifying the option, the item is displayed as a blank field.
5. No items after an end are displayed.
   (Example) In the case of N1 → R1 (No.), N2 → R2 (Type Number), N 3 → R3 (MG), N4 → R-1 (End), N5 → R4 (Pot), only the items "No., Type Number, and MG" are displayed on the screen.
6. If G10 L77 P3 is specified, the setting immediately before is initialized.
7. If an invalid value is set in R (tool management data display item), a blank field is displayed.

10.3.2 Setting of Spindle Position / Standby Position Display

In MG on the tool management data screen, a spindle position or standby position is displayed as a number such as 11, 12, and 13. With the spindle position/standby position display setting function, three arbitrary characters can be displayed using the G10 format.

Format

```
G10L77P4;
N_;
P_R_;
G11;
  N_: Spindle position/standby position setting
  P_: Character number
  R_: Character code
```

Explanation

- Spindle position/standby position setting (N_)

Specify a spindle position or standby position to be renamed. The table below indicates the values to be specified.

	Spindle position			
	First	**Second**	**Third**	**Fourth**
1st path	111	112	113	114
2nd path	211	212	213	214
3rd path	311	312	313	314
4th path	411	412	413	414
5th path	511	512	513	514
6th path	611	612	613	614
7th path	711	712	713	714
8th path	811	812	813	814
9th path	911	912	913	914
10th path	1011	1012	1013	1014

	Standby position			
	First	**Second**	**Third**	**Fourth**
1st path	121	122	123	124
2nd path	221	222	223	224
3rd path	321	322	323	324
4th path	421	422	423	424
5th path	521	522	523	524
6th path	621	622	623	624
7th path	721	722	723	724
8th path	821	822	823	824
9th path	921	922	923	924
10th path	1021	1022	1023	1024

- Character number (P_)

Specify a character number (1 to 3). Up to three characters are displayed. If a character string to be specified is shorter than three

characters, specify 0 in the leading blank character position(s). A character string immediately before 0 is displayed.

- Character code (R_)

Set the name of a spindle position/standby position by using a character code (ASCII code or Shift JIS code).

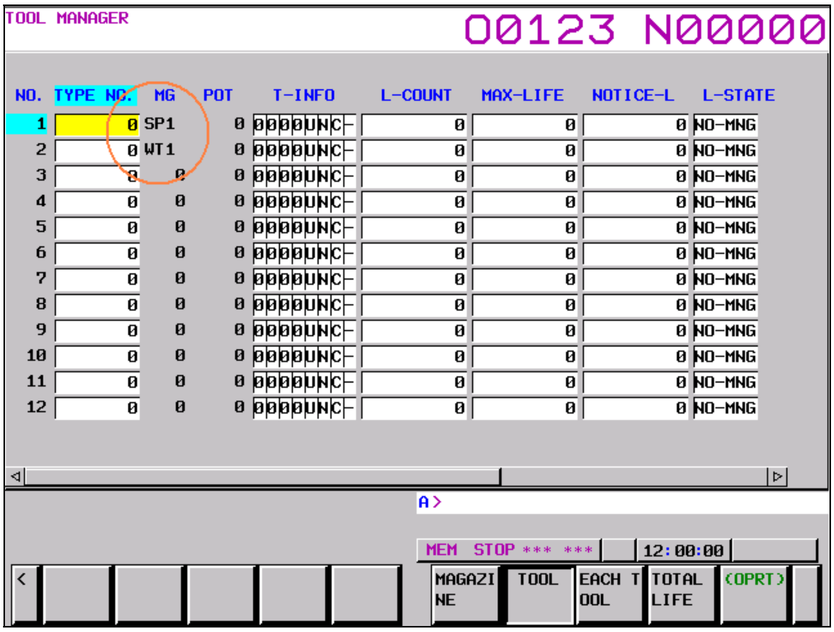
- For a character code, use ASCII code or Shift JIS code.
- No character code data check is made.

Example

When spindle 1 is named "SP1", and standby 1 is named "WT1"

G10L77P4;	Set spindle position/standby position
N111;	Specify spindle 1
P1 R83;	53h as ASCII code for "S"
P2 R80;	50h as ASCII code for "P"
P3 R49;	31h as ASCII code for "1"
N121;	Specify standby 1
P1 R87;	57h as ASCII code for "W"
P2 R84;	54h as ASCII code for "T"
P3 R49;	31h as ASCII code for "1"
G11;	Cancel the setting mode

In the MG item on the tool management data screen, spindle 1 is displayed as "SP1", and standby 1 is displayed as "WT1".



NOTE
Data registered becomes effective after the screen display is switched to the tool management screen.

10.3.3 Input of Customize Data with the Decimal Point

With the function for input of customize data with the decimal point, the number of decimal places can be set using the G10 format for each customize data item (customize data 1, ..., 40) to enable data input with the decimal point.

Format

G10L77P5;
N_R_;
G11;
N_: Customize data number
R_: Decimal point position

Explanation

- Customize data number (N_)

- A value of 1 to 4 can be specified for N.
- When tool management function customize data extension (5 to 20) is available, a value of 1 to 20 can be specified.
- When tool management function customize data extension (5 to 40) is available, a value of 1 to 40 can be specified.

- Decimal point position (R_)

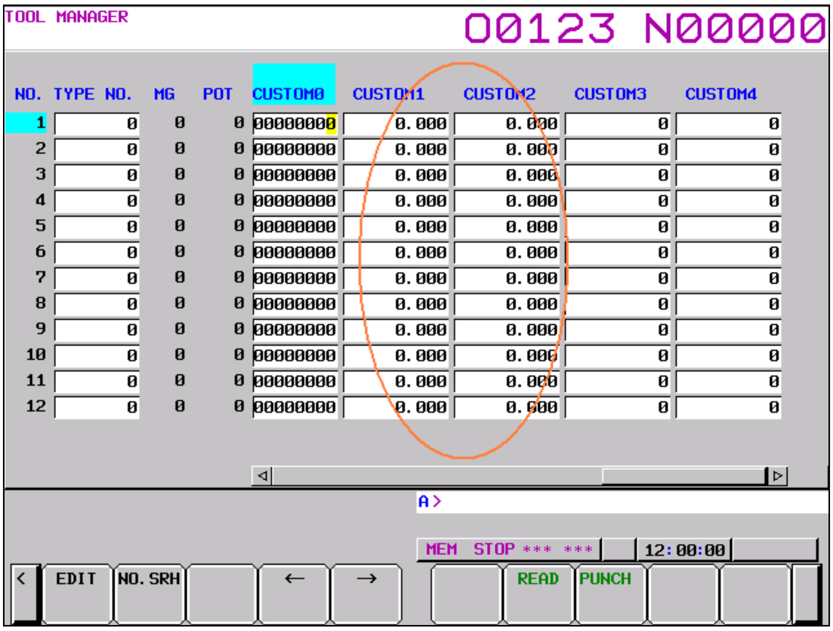
Specify a decimal point position (0 to 7). When 0 is set, the number of decimal places of customize data cannot be input.

NOTE
1 If G10 L77 P5 is terminated normally, the power must be turned off before operation is continued.
2 The setting becomes effective after the power is turned off then back on.
3 When the number of decimal places is set for customize data, the number of decimal places can be input only through the MDI keys.
4 To set the number of decimal places for customize data 5 to 20, the customize data extension (5 to 20) option for the tool management function or the customize data extension (5 to 40) option for the tool management function is required.
5 To set the number of decimal places for customize data 21 to 40, the customize data extension (5 to 40) option for the tool management function is required.

Example 1

When customize data 1 and customize data 2 are input with three decimal places

G10L77P5;	Set the number of decimal places for customize data
N1 R3;	Set the number of decimal places to 3 for customize data 1
N2 R3;	Set the number of decimal places to 3 for customize data 2
G11;	Cancel the setting mode



- Use in the G10 format

Between G10 and G11, the decimal point cannot be specified at any address. So, if decimal point input for customize data is enabled with this function, a G10 L75/L76/L77 command using a custom macro variable (#8431 to #8450) reads and sets the value of the integer part.

Example 2

(Example 1)

Condition:

"3" is set as the decimal point position of customize data 1.
"1" is set as the decimal point position of customize data 2.

Operation:

Data is transferred from customize data 1 to customize data 2 by using a custom macro variable.

G10L77P5;	<1> Set customize data decimal point position
N1 R3;	<2> Set 3 as decimal point position of customize data 1
N2 R1;	<3> Set 1 as decimal point position of customize data 2
G11;	<4> Cancel the setting mode
;	
G10 L75 P1;	<5> Register tool management data
N01;	<6> Register with No. 1
P1 R12345;	<7> Set ”12.345” for customize data 1
P2 R#8431;	<8> Set ”1.2” for customize data 2
G11;	<9> Cancel the setting mode

In Example 1, customize data 1 is directly set in customize data 2 by using a custom macro variable.
Customize data 1 holds "12.345". In step <8>, only the integer part is read and processed as "P2 R12".
So, "1.2" is set in customize data 2.

(Example 2)

Condition:

"3" is set as the decimal point position of customize data 1.
"1" is set as the decimal point position of customize data 2.

Operation:

By using the custom macro variable of customize data 1, the data is transferred to customize data 2 after being multiplied by 1000.

G10L77P5;	<1> Set customize data decimal point position
N1 R3;	<2> Set 3 as decimal point position of customize data 1
N2 R1;	<3> Set 1 as decimal point position of customize data 2
G11;	<4> Cancel the setting mode
;	
G10 L75 P1;	<5> Register tool management data
N01;	<6> Register with No. 1
P1 R12345;	<7> Set ”12.345” for customize data 1
P2 R[#8431*1000];	<8> Set ”1234.5” for customize data 2
G11;	<9> Cancel the setting mode

In Example 2, customize data 1 is directly set in customize data 2 by using a custom macro variable.
Customize data 1 holds "12.345". In step <8>, the data is multiplied by 1000 to eliminate the fractional part.
So, this command is equivalent to "P2 R12345", so that "1234.5" is set in customize data 2.

10.3.4 Protection of Various Tool Information Items with the KEY Signal

When tool management data is in the edit state, various information items can be modified. By setting bit 0 of parameter No. 13204 to 1, tool management data can be protected with the KEY signal so that various information items are not registered, modified, and deleted.

10.3.5 Selection of a Tool Life Count Period

A tool life count period can be chosen between 1 sec and 8 msec on a tool-by-tool basis.

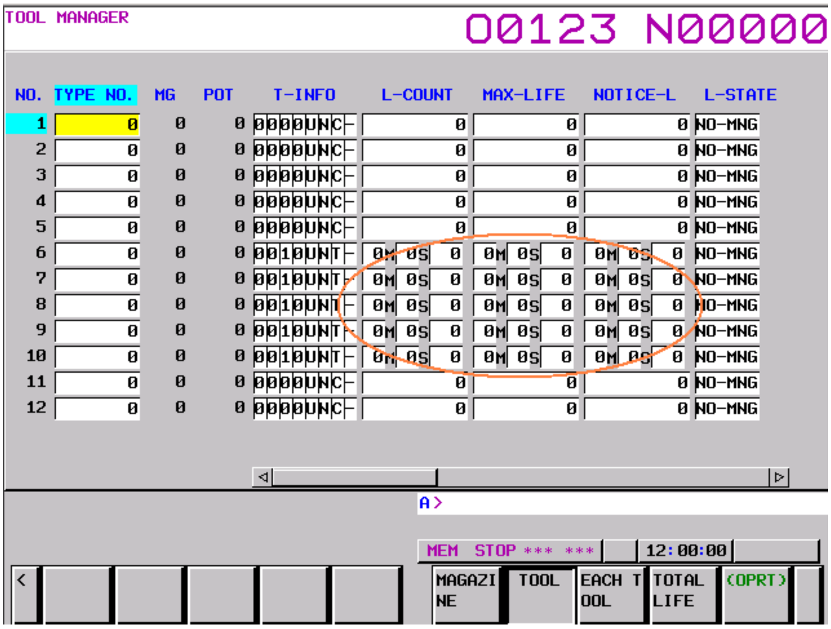
- Tool life count period selection

Bit 5 of tool information is used to make a life count period selection.

Item	Description
Data length	1 byte (flag data)
#5 REV	0: A life count period of 1 sec is used. 1: A life count period of 8 sec is used.

Range of count is as follows.
1sec: 0 to 3,599,999 seconds (999 hours 59 minutes 59 seconds)
8msec: 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

NOTE
This function is valid when the tool information TIM (#1) is set to 1.



10.3.6 Individual Data Screen

All data for a specified tool can be extracted and displayed.

10.3.7 Total Life Time Display for Tools of The Same Type

The remaining lives of tools with the same type numbers are totaled, and totals are displayed in order by tool type number or by remaining life. Also, data of tools with the same tool type numbers are displayed in a list.

10.4 TOOL MANAGEMENT FUNCTION OVERSIZE TOOLS SUPPORT

Overview

Tool management function oversize tools support is added to the tool management function.
The figure of an oversize tool can be defined freely, and the figure of each oversize tool is registered. When an oversize tool is stored in a cartridge, interference with tools stored in other pots is considered. This function is usable with cartridges of chain type and matrix type.

Format

By G10 data input, an oversize tool number can be set in the tool management data. Moreover, tool figure data can be set.

<Registration of new tool management data>

```
G10 L75 P1 ;
N_;
A_;
G11 ;
  N_: Tool management data number
  A_: Specify tool figure number (0 to 20).
```

<Modification of tool management data>

```
G10 L75 P2;
N_;
A_;
G11 ;
  N_: Tool management data number
  A_: Specify tool figure number (0 to 20).
```

<Registration of tool figure data>

```
G10 L77 P6 ;
N_P_Q_R_S_T_;
G11 ;
  N_: Tool figure number
  P_: Number of pots to be occupied in left direction (in steps
      of 0.5 pot)
  Q_: Number of pots to be occupied in right direction (in
      steps of 0.5 pot)
  R_: Number of pots to be occupied in upper direction (in
      steps of 0.5 pot)
  S_: Number of pots to be occupied in lower direction (in
      steps of 0.5 pot)
  T_: 0 for figure A, or 1 for figure B
```

NOTE

1. If a target tool is registered in a cartridge and interferes with other tools in registration or modification of tool figure data of the tool management data, PS alarm 5360 is issued. (The data is not input.)
2. If a tool interferes with other tools in registration or modification to the cartridge management table, PS alarm 5360 is issued. (The data is not input.)
3. If an attempt is made to modify tool figure data, and the tool for which the tool figure data number subject to modification is set is registered in the cartridge, PS alarm 5360 is issued. (The data is not input.)

11 AUXILIARY FUNCTION

Overview

There are two types of auxiliary functions ; auxiliary function (M code) for specifying spindle start, spindle stop program end, and so on, and secondary auxiliary function (B code) for specifying index table positioning.

When a move command and auxiliary function are specified in the same block, the commands are executed in one of the following two ways:

(1) Simultaneous execution of the move command and auxiliary function commands.

(2) Executing auxiliary function commands upon completion of move command execution.

The selection of either sequence depends on the machine tool builder's specification. Refer to the manual issued by the machine tool builder for details.

11.1 AUXILIARY FUNCTION (M FUNCTION)

When a numeral is specified following address M, code signal and a strobe signal are sent to the machine. The machine uses these signals to turn on or off its functions.
Usually, only one M code can be specified in one block. In some cases. Depending on the setting of bit 7 (M3B) of parameter No. 3404, up to three M codes can be specified.
Which M code corresponds to which machine function is determined by the machine tool builder. The machine processes all operations specified by M codes except those specified by M98, M99,M198 or called subprogram(Parameter No.6071 to 6079), or called custom macro (Parameter No.6080 to 6089). Refer to the machine tool builder's instruction manual for details.

Explanation

The following M codes have special meanings.

- M02,M03 (End of program)

This indicates the end of the main program.
Automatic operation is stopped and the CNC unit is reset. (This differs with the machine tool builder.)
After a block specifying the end of the program is executed, control returns to the start of the program.
Parameter M02 (No. 3404#5) or parameter M30 (No. 3404#4) can be used to disable M02, M30 from returning control to the start of the program.

- M00 (Program stop)

Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation. (This differs with the machine tool builder.)

- M01 (Optional stop)

Similarly to M00, automatic operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel has been pressed.

- M98 (Calling of subprogram)

This code is used to call a subprogram. The code and strobe signals are not sent. See the subprogram II-13.3 for details.

- M99 (End of subprogram)

This code indicates the end of a subprogram.
M99 execution returns control to the main program. The code and strobe signals are not sent. See the subprogram section II-13.3 for details.

11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

Usually, only one M code can be specified in one block. By setting bit 7 (M3B) of parameter No. 3404 to 1, however, up to three M codes can be specified simultaneously in one block.
Up to three M codes specified in one block are output to the machine at the same time. So, when compared with a case where a single M code is specified in one block, a reduced machining cycle time can be achieved.

Explanation

CNC allows up to three M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.
M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.
Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block.
Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks. Meanwhile, multiple of M codes that direct the CNC only to send the M codes themselves (without performing internal operations) can be specified in a single block.
However, it is possible to specify multiple M codes that are sent to the machine in the same block unless they direct the CNC to perform internal operations. (Since the processing method depends on the machine, refer to the manual of the machine tool builder.)

Example

One M command in a single block	Multiple M commands in a single block
M40; M50; M60; G28G91X0Y0Z0; : : :	M40M50M60; G28G91X0Y0Z0; : : : : :

11.3 M CODE GROUPING FUNCTION

Overview

Classifying a maximum of 500 M codes into a maximum of 127 groups allows the user:

- To receive an alarm if an M code that must be specified alone is included when multiple M codes are specified in a block.
- To receive an alarm if M codes belonging to the same group are specified in the same block when multiple M codes are specified in a block.

11.3.1 Setting an M Code Group Number Using the Setting Screen

- Procedure for displaying the M code group setting screen

You can use the "M code group setting screen" to set a group number for each M code.

Display the "M code group setting screen" using the following procedure:

(1) Press function key SYSTEM and the continuous menu key several times. Soft key [M CODE] appears.
(2) Press soft key [M CODE].

In the "NUMBER" field, M codes for which an M code group can be set are displayed.
An M code group can be set for the following M codes: M00 to M99, and any 400 M codes selected from M100 and subsequent M codes. For details of how to add the 100th and subsequent M codes, see the explanation of parameters Nos. 3441 to 3444.

In the "VALUE" field, the M code group number corresponding to each M code is displayed.

· Setting a group number

To set an M code group number on the "M code group setting screen," use he following procedure:

1 Select the MDI mode.
2 Set "PARAMETER WRITING" on the setting screen to 1.
3 Display the "M code group setting screen.
4 Move the cursor to the M code to be set using page keys and cursor keys. You can also enter the number of the M code to be set and press soft key [NO.SRH] to move the cursor to the M code.
5 Enter a group number and press soft key [INPUT] or the INPUT key.

The valid range of M code group numbers is from 1 to 127 (127 groups). If a value of 0 is input, it is not registered as an M code group.

- Examples of setting parameters Nos. 3441 to 3444

In the following examples, the number of digits of an M code is 4.
<1> to <4> indicate parameters Nos. 3441 to 3444.

(1) When <1> = 300, <2> = 400, <3> = 500, and <4> = 900 are set

Number
0000
: 100 codes
0099
0300
: 100 codes
0399
0400
: 100 codes
0499
0500
: 100 codes
0599
0900
: 100 codes
0999

M code groups can be set for M0000 to M0099, M0300 to M0599, and M0900 to M999.
M codes M0300 to M0599 and M0900 to M999 are added to the M code group setting screen.

(2) When <1> = 200, <2> = 0, <3> = 550, and <4> = 800 are set

Number
0000
:
0099
0200
:
0299
0550
:
0649
0800
:
0899

M code groups can be set for M0000 to M0099, M0200 to M0299, M0550 to M0649, and M0800 to M0899.
(The setting of parameter <2> is invalid because it is 0.)
In this case, M codes M0200 to M0299, M0550 to M0649, and M0800 to M0899 are added to the M code group setting screen.

11.3.2 Setting an M Code Group Number Using a Program

You can execute a program to set an M code group number and M code name. The command format is shown below.

Format

G10 L40 Pn Rg ;

Pn: "n" specifies an M code.

Rg: "g" specifies an M code group number.

NOTE

1. If the format is invalid, an alarm (PS1144) is issued.
2. If an M code group cannot be set for the M code specified for the P command or if the group number specified for the R command is not within the range between 0 and 127, an alarm (PS1305) is issued.

(Example)

Executing the following program sets M code group "1" to M03:

```
G10   L40   P03   R1 ;
```

11.3.3 M Code Group Check Function

When multiple M commands in a single block (enabled when bit 7 (M3B) of parameter No. 3404 is set to 1) are used, you can check the following items. You can also select whether to check the items using bit 1 (MGC) of parameter No. 3400.

(1) M code to be specified in a single block containing no other M codes
If an M code which must be specified in a single block containing no other M codes is specified together with another M code, an alarm (PS5016) is issued.
(2) M codes in the same group
If multiple M codes in the same group are specified together, an alarm (PS5016) is issued.

The valid range of group numbers is from 0 to 127 (128 groups).
Group numbers 0 and 1 have special meaning. Note the following points:

- Each M code with group number 1 is assumed to be an M code to be specified in a single block containing no other M codes.
- For each M code with group number 0, the “same group M code check” is ignored. That is, when multiple M codes with group number 0 are specified in a single block, the alarm is not issued.
- For each M code with group number 0, the “check for an M code to be specified in a single block containing no other M codes” is not ignored. That is, if an M code with group number 1 and an M code with group number 0 are specified in a single block, the alarm is issued.
- For M codes that are not to be output to the machine such as M98, M99, M198, and M codes for subprogram and macro calls (set in parameters Nos. 6071 to 6079 and Nos. 6080 to 6089 and with the macro executor), be sure to set 0 as the group number.
- For M00, M01, M02, M30, and M codes for which buffering is suppressed (set in parameters Nos. 3411 to 3432), be sure to set 1 as the group number.

11.4 SECOND AUXILIARY FUNCTIONS (B CODES)

Overview

If a value with a maximum of eight digits is specified after address B, the code signal and strobe signal are transferred for calculation of the rotation axis. The code signal is retained until the next B code is specified.
Only one B code can be specified for each block. When the maximum number of digits are specified by parameter No.3033, an alarm is issued if the number of digits of a command exceeds the specified number.
In addition, the address used for specifying the second auxiliary function can be changed to an address other than address B (address A, C, U, V, or W) by setting parameter No.3460.
However, the address used for the second auxiliary function cannot also be used as the address of the controlled axis. For details, refer to the manual available from the machine tool builder.

Explanation

- Range of specification

-99999999 to 99999999

- Output value

The value specified after the address of the second auxiliary function is output on the code signals B00 to B31. Note the following about a output value.

1. When a command with a decimal point or a negative command is disabled
   (When parameter AUP (No.3450#0) is set to 0)

   When the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is, regardless of the desktop calculator decimal point setting (parameter DPI (No.3401#0)).
   Example:

Specified value	Output value
B10	10

   When the second auxiliary function with a decimal point is specified, alarm PS0007 is issued.
   When the second auxiliary function is specified with a negative value, alarm PS0006 is issued.

2. When a command with a decimal point or a negative command is enabled
(When parameter AUP (No.3450#0) is set to 1)

When the desktop calculator decimal point setting is not specified (when parameter DPI (No.3401#0) is set to 0), if the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is.
Example:

Specified value	Output value
B10	10

When desktop calculator decimal point input is specified (when parameter DPI (No.3401#0) is set to 1), if the second auxiliary function with no decimal point is specified, the specified value multiplied by a magnification is output on the code signals. (Magnifications are shown in Table 11.4 (a).)
Example:

Specified value	Output value	
B10	10000	(When metric input is used and the reference axis is IS-B. The magnification is 1000.)

When the second auxiliary function with a decimal point is specified, the specified value multiplied by a magnification is output to the code signals. (Magnifications are shown in Table 11.4 (a).)
Example:

Specified value	Output value	
B10.	10000	(When metric input is used and the reference axis is IS-B. The magnification is 1000.)
B0.123	1230	(When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

The magnification is determined as shown below according to the setting unit of the reference axis (specified by parameter No.1031) and parameter AUX (No.3405#0).

Table 11.4 (a) Magnifications for an output value when the second auxiliary function with a decimal point is specified for desktop calculator decimal point input

Setting unit		Parameter AUX = 0	Parameter AUX = 1
Metric input system	Reference axis: IS-A	100×	100×
	Reference axis: IS-B	1000×	1000×
	Reference axis: IS-C	10000×	10000×
	Reference axis: IS-D	100000×	100000×
	Reference axis: IS-E	1000000×	1000000×
Inch input system	Reference axis: IS-A	100×	1000×
	Reference axis: IS-B	1000×	10000×
	Reference axis: IS-C	10000×	100000×
	Reference axis: IS-D	100000×	1000000×
	Reference axis: IS-E	1000000×	10000000×

⚠ CAUTION

If a decimal fraction remains after multiplying the specified value with a decimal point by a magnitude in Table 11.4 (a), the fraction is truncated.
Example:

Specified value	Output value
B0.12345	1234 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

NOTE

If the number of digits of the specified value exceeds the allowable number of digits (set by parameter No.3033), alarm PS0003 is issued. When the specified value is multiplied by a magnitude in Table 11.4 (a), the allowable number of digits must be set for the resultant value.

Limitation

Addresses used for the second auxiliary functions (addresses specified with B or parameter No. 3460) cannot be used as the addresses used for controlled axis names.

12 PROGRAM MANAGEMENT

12.1 FOLDERS

Overview

Folders can be created in program memory.

12.1.1 Folder Configuration

The following folders can be created:

- Folder names are up to 32 characters long.
- The following characters can be used in folder names:
  Alphabetical characters (uppercase and lowercase letters), numeric characters, and the symbols below:
  – + _ .
  Because "." and ".." are reserved folder names, they cannot be used.

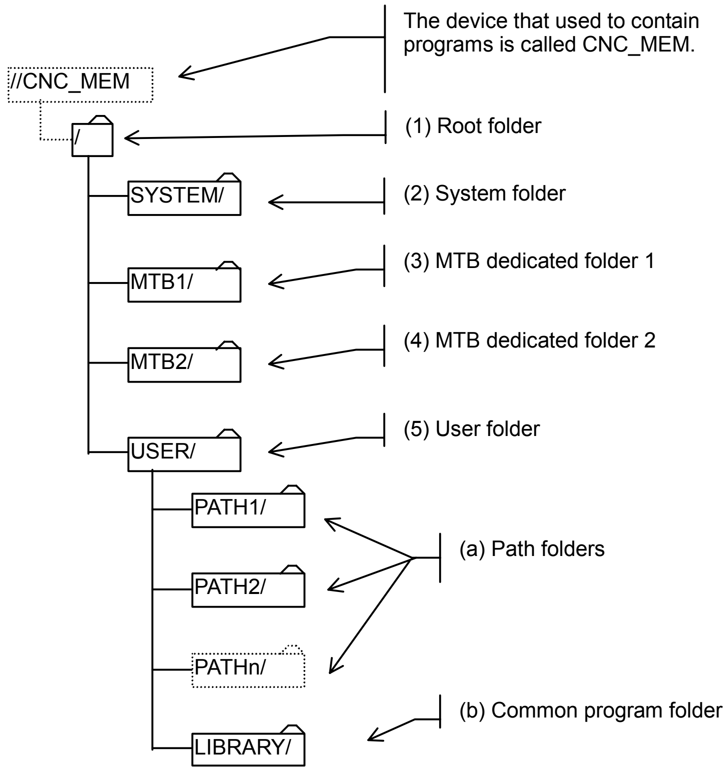
- Initial folders

When program memory is initialized, folders having predefined structures and names are created. These folders are called initial folders.

(1) Root folder
The parent folder of all folders
(2) System folder
Contains subprograms and macro programs of the system.
(3) MTB dedicated folder 1
Contains subprograms and macro programs created by the machine tool builder.
(4) MTB dedicated folder 2
Contains subprograms and macro programs created by the machine tool builder.
(5) User folder
Contains programs created by the user.
The following folders are further created in this folder:
(a) Path folders (As many folders as the number of paths are created.)
Each contains main programs, subprograms, and macro programs used for the corresponding path.
(b) Common program folder
Contains subprograms and macro programs used commonly.

> **NOTE**
> The initial folders can neither be deleted nor renamed.

[Initial folder configuration]



- User created folders

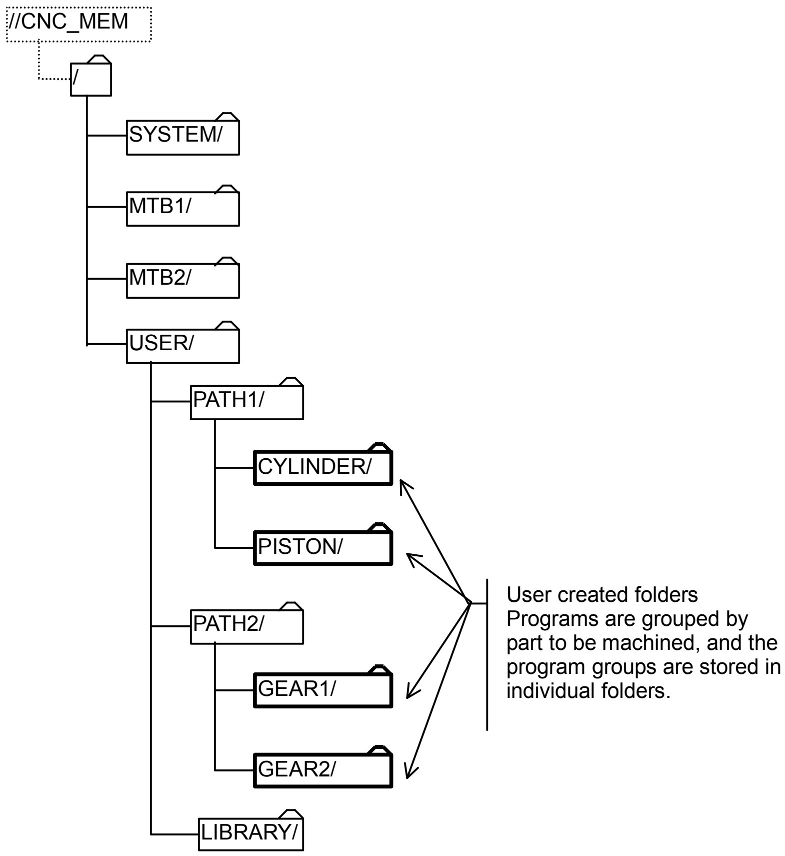
Folders other than the initial folders are called user created folders.
User created folders can be created in the following initial folders:

- User folder
- Path folders

User created folders can contain user created main programs, subprograms, and macro programs.

> **NOTE**
> 1 Folder names must each be unique in the same folder.
> 2 Each time a user created folder is created, the number of programs that can be registered decreases by one.
> 3 Folder hierarchical levels of user created folders are limited.
> Up to three hierarchical levels starting from the user folder (USER/) are permitted.

[Sample folder configuration]



12.1.2 Folder Attributes

The following attributes can be set for folders except the root folder:

- Edit disable
- Edit/display disable

- Edit disable

Editing of the programs and folders in a folder can be disabled.
A program in the folder can be output to an external device.
A program cannot be input (registered) from an external device to the folder.

- Edit/display disable

Editing and display of the programs and folders in a folder can be disabled.
When this attribute is set for a folder, the programs and folders in the folder become invisible. (The folder appears to contain nothing.)
A program in the folder cannot be output to an external device and a program cannot be input (registered) from an external device to a folder.

12.1.3 Default Folders

Default folders are folders on which operations are performed when no folder is specified. There are two types of default folders as follows:

- Foreground default folder
- Background default folder

- Foreground default folder

A folder used for foreground operations except automatic operations and program editing is set.
The target operations include:

- Program input/output
- External data input
- External workpiece number search

- Background default folder

A folder used for background operations is set.
The target operations include:

- Program input/output
- External I/O device control

NOTE

1. When the foreground or background default folder is not set, a path folder, which is an initial folder, is assumed.
2. Foreground and background default folder settings are stored in default folder setting files.
3. When a program file, program folder, or program folder management file is cleared, the default folder setting file is cleared at the same time.

12.2 FILES

Overview

Desired file names can be given to part programs in program memory.

12.2.1 File Name

File names can be set as follows:

- File names are up to 32 characters long.
- The following characters can be used in file names:
  Alphabetical characters (uppercase and lowercase letters), numeric characters, and the symbols below:
  – + _ .
  Because "." and ".." are reserved file names, they cannot be used.

- File names and program numbers

File names are associated with program numbers as explained below.
When the file name of a program consists of "O" plus a numeric with the following limitation, the program can be handled also by program number.

- The numeric must be a leading zero suppressed value from 1 to 9999.

When the file name of a program does not have the above format, the program cannot be handled by program number.
When a file name consisting of "O" plus a numeric does not satisfy the above limitation, the file cannot be created.

Example)

File names that can be treated as program numbers

O123	Program number 123
O1	Program number 1
O3000	Program number 3000
O9999	Program number 9999

File names that cannot be treated as program numbers

ABC
o123
O123.4

NOTE
1. File names must each be unique in the same folder.
2. When the file name of a program is not treated as a program number, the program is restricted as follows:
   - The program cannot be specified by program number.
   - Information output by program number is impossible..

- Displaying File Names and Program Numbers

The file name of the program selected or being executed as the main program is displayed as shown in Figs. 12.2.1 (a) to 12.2.2 (c).

- For file names that can be handled as program numbers, the program number is displayed.

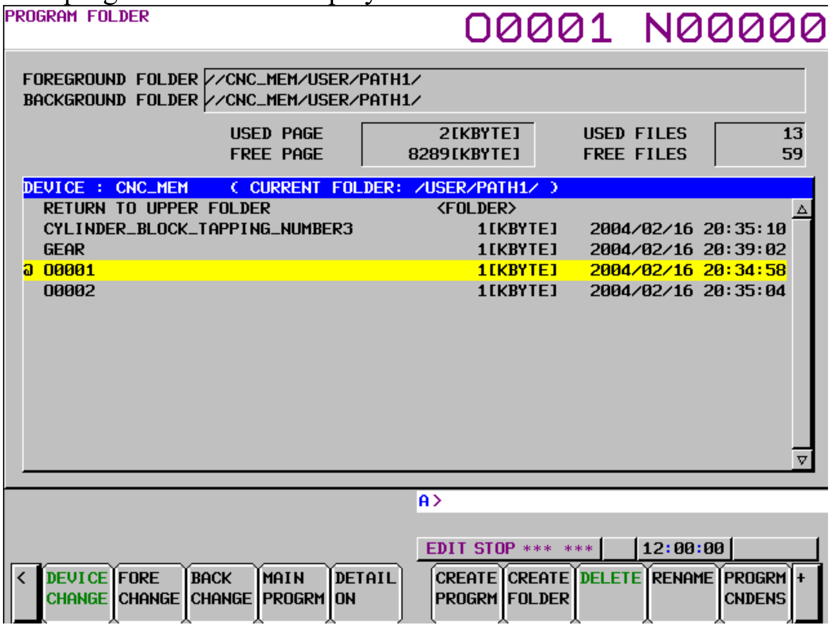


Fig. 12.2.1 (a) File name display 1

- For file names that cannot be handled as program numbers, the file name is displayed.
  The display layout varies as shown below, depending on the length of a file name.

Fig. 12.2.1 (b) File name display 2

Fig. 12.2.1 (c) File name display 4

12.2.2 File Attributes

The following attributes can be set for files:

- Edit disable
- Edit/display disable
- Encoding
- Change protection level/output protection level

- Edit disable

Editing of a specified program can be disabled.
A program cannot be input (registered) from an external device to the folder.

- Edit/display disable

Editing and display of a specified program can be disabled.
When this attribute is set, the program becomes invisible.
(The folder appears not to contain this program.)
An output to an external device and an input from an external device (program registration) cannot be made.

- Encoding

A specified program can be encoded.
For details of encoding, see the description of the “Program encryption” function.
An output to an external device and an input from an external device (program registration) cannot be made.

- Change protection level/output protection level

With the 8-level data protection function, change and output protection can be provided for a specified program.
For details of the 8-level data protection function, see the description of the “Protection of data at eight levels” function.

12.3 RELATION WITH CONVENTIONAL FUNCTIONS

This section explains relation with conventional functions when folder names and file names are used.

12.3.1 Relation with Folders

This subsection explains how folders are used for operations and editing.

- Automatic operation
Main program

As the main program running for an automatic operation, a program in a desired folder can be selected.

Subprogram (called by M98/G72.1/G72.2)
Macro program (called by G65/G66/G66.1/M96)

- Subprogram call (M98)
- Macro call (simple call G65/modal call G66, G66.1)
- Macro interrupt (M96)
- Figure copy (G72.1, G72.2)

When a call listed above is made, folders are searched in the following order, and the program found first is called:
<1> Folders containing the main program
<2> Common program folder, which is an initial folder

Bit 7 (SCF) of parameter No. 3457 can be used to add the following search folders. (The folders are searched in the order listed below.) The folder to be actually enabled is separately set by bits 0 to 3 of parameters No. 3457.
<3> MTB dedicated folder 2, which is an initial folder
<4> MTB dedicated folder 1, which is an initial folder
<5> System folder, which is an initial folder

Subprogram (called by M code/specific address/2nd auxiliary function)
Macro program (called by G code/M code/T code/one touch macro)

- Subprogram call by M code/specific address/2nd auxiliary function code
- Macro call by G code/M code/T code
- One touch macro call

For programs called as listed above, the folders to be searched are set in the parameters (No.3457#0 to #3) in advance. (The search order is described below.)
The folders set as the search targets are searched, and the program found first is called.
<1> Common program folder, which is an initial folder
<2> MTB dedicated folder 2, which is an initial folder
<3> MTB dedicated folder 1, which is an initial folder
<4> System folder, which is an initial folder

- Program editing

A program in any folder can be edited.

- Program I/O

The following functions are performed for default folders:

- Program input from external devices
- Program output to external devices
  (Except the format with folder names)

The following functions are performed for a set background default folder:

- External I/O device control

- External data input

A set foreground default folder is searched for an external program.

- External workpiece number search

A set foreground default folder is searched for an external workpiece number.

- Macro executor

Programs called by execution macros, conversational macros, and auxiliary macros are programs in P-Code files regardless of the default folder.
For programs on the macro executor, the folder function and file name function are unavailable.

12.3.2 Relation with File Names

File names can be used with the following functions:

- Subprogram call (M98)
- Macro call (simple call G65/modal call G66, G66.1)
- Interruption type macro call (M96)
- Subprogram call in figure copy (G72.1, G72.2)
- Program I/O with external devices

- Subprogram call by file name
- Macro call by file name

- Subprogram call (M98)
- Macro call (G65/G66/G66.1)
- Interruption type macro call (M96)
- Figure copy (G72.1, G72.2)

When a program is called in the above functions, a subprogram call by file name and a macro call by file name can be used.

- Subprogram call by file name
  M98 <file-name> Lxx ;
- Macro call by file name
  G65 <file-name> Lxx argument-specification ;
  G66 <file-name> Lxx argument-specification ;
  G66.1 <file-name> Lxx argument-specification ;
- Interruption type macro call
  M96 <file-name> ;
- Subprogram call by file name in figure copy (G72.1, G72.2)
  G72.1 <file-name> Lxx Xxx Yxx Rxx ;
  G72.2 <file-name> Lxx Ixx Jxx ;

In the above subprogram and macro calls, the program having the file name indicated by <file-name> is called.

Format examples:

- Subprogram call
  M98 <R50> L1 ;
- Macro call
  G65 <R50> L1 A0 ;
  G66 <R50> L1 A1 ;
  G66.1 <R50> L1 A2 ;
- Interruption type macro call
  M96 <R50> ;
- Subprogram call in figure copy
  G72.1 <R50> L1 X0 Y0 R0 ;
  G72.2 <R50> L1 I0 J0 ;

NOTE

1. When characters in <> are read, they are treated in the same way as for characters in comments. So, note that these characters are treated differently from other significant information portions.
   Refer to Appendix B “PROGRAM CODE LIST” for detils.
2. The <file-name> word must be placed immediately after each word for calling (M98, G65, and so forth).

12.3.3 Related Parameters

This subsection lists the meanings of parameters related to program numbers and the folders and programs to be manipulated or executed.

Parameter No.	Bit No.	Description	Manipulation/execution target
3202	0 (NE8)	Disables or enables editing of programs O8000 to O8999.	Corresponding programs in all folders
	4 (NE9)	Disables or enables editing of programs O9000 to O9999.	Same as above
3204	3 (P8E)	Disables or enables editing of programs O80000000 to O8999999.	Same as above
	4 (P9E)	Disables or enables editing of programs O90000000 to O99999999.	Same as above
	5 (SPR)	Assumes or does not assume a particular program number in the nine thousands to be a number obtained by adding 90000000.	Same as above
3210/3211	-	Password/keyword for protecting programs in the nine thousands	Same as above
3222/3223	-	Program protection range (minimum value)/(maximum value)	Same as above
3404	2 (SBP)	In the subprogram call function, address P in the M198 block specifies a file/program number.	(Not dependent on the folder function and file name function)
6001	5 (TCS)	Calls or does not call a custom macro by T code.	The following initial folders can be selected by parameter setting: - Common program folder - MTB dedicated folder 2 - MTB dedicated folder 1 - System folder
6050 to 6059	-	G code for calling a custom macro with program No. 9010 to 9019	Same as above
6071 to 6079	-	M code for calling a subprogram with program No. 9001 to 9009	Same as above
6080 to 6089	-	M code for calling a custom macro with program No. 9020 to 9029	Same as above
6090/6091	-	ASCII code for calling a subprogram with program No. 9004/9005	Same as above
8341/8343	-	Target program number (for 4- and 8-digit O number) for sequence number comparison and stop	Foreground or background default folder depending on the mode

13 PROGRAM CONFIGURATION

Overview

- Main program and subprogram

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

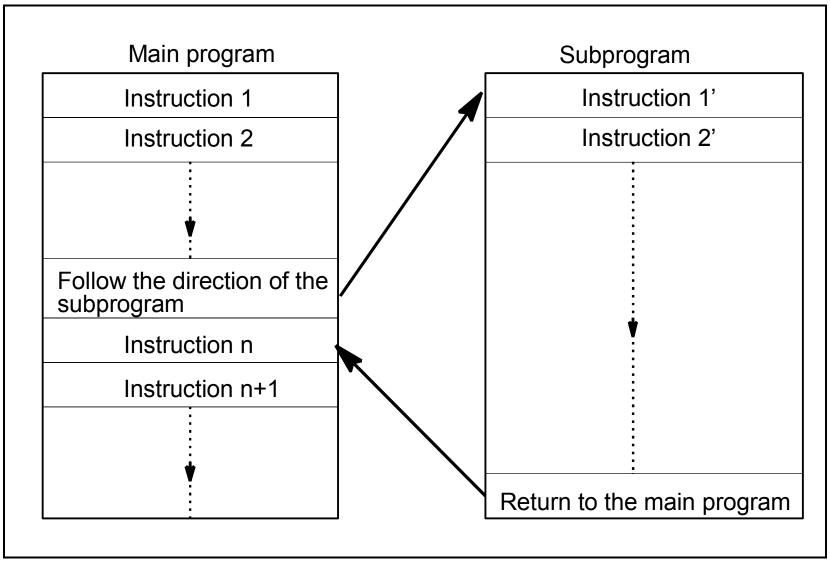


Fig. 13 (a) Main program and Subprogram

The CNC memory can hold up to 1000 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine. See III-9 or III-10.4 for the methods of registering and selecting programs.

- Program components

A program consists of the following components:

Table 13 (a) Program components

Components	Descriptions
Program code start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Program code end	Symbol indicating the end of a program file

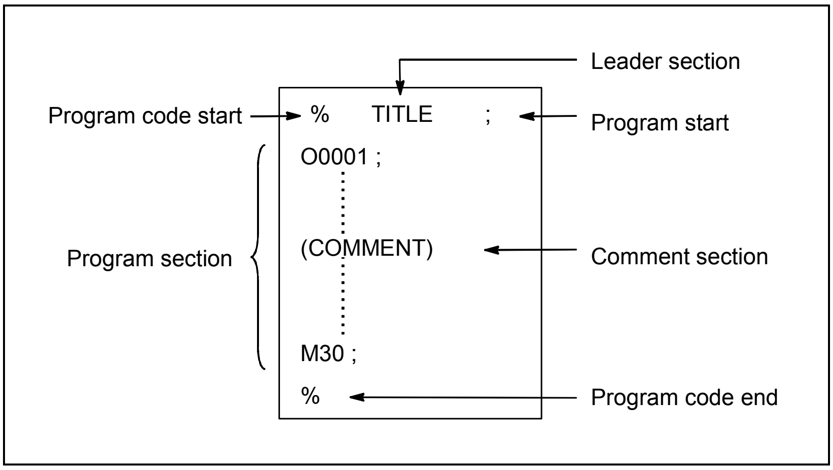


Fig. 13 (b) Program configuration

- Program section configuration

A program section consists of several blocks. A program section starts with a program number or file name and ends with a program end code.

Program section configuration	**Program section**
Program number	O0001 ;
Block 1	N1 G91 G00 X120.0 Y80.0 ;
Block 2	N2 G43 Z-32.0 H01 ;
:	:
Block n	Nn Z0 ;
Program end	M30 ;

A block contains information necessary for machining, such as a move command or coolant on/off command. Specifying a slash (/) at the start of a block disables the execution of some blocks (see "optional block skip" in II-13.2).

13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See II-13.2 for a program section.

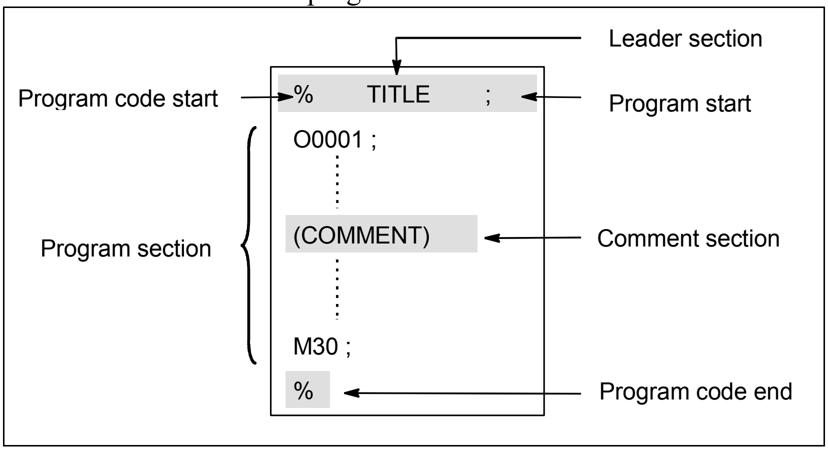


Fig. 13.1 (a) Program configuration

Explanation

- Program code start

The program code start indicates the start of a file that contains NC programs.
The mark is not required when programs are entered using SYSTEM P or ordinary personal computers. The mark is not displayed on the screen. However, if the file is output, the mark is automatically output at the start of the file.

Table 13.1 (a) Code of a program code start

Name	ISO code	EIA code	Notation in this manual
Program code start	%	ER	%

- Leader section

Data entered before the programs in a file constitutes a leader section.
When machining is started, the label skip state is usually set by turning on the power or resetting the system. In the label skip state, all information is ignored until the first end-of-block code is read. When a file is read into the CNC unit from an I/O device, leader sections are skipped by the label skip function.
A leader section generally contains information such as a file header. When a leader section is skipped, even a TV parity check is not made. So a leader section can contain any codes except the EOB code.

- Program start

The program start code is to be entered immediately after a leader section, that is, immediately before a program section.
This code indicates the start of a program, and is always required to disable the label skip function.
With SYSTEM P or ordinary personal computers, this code can be entered by pressing the return key.

Table 13.1 (b) Code of a program start

Name	ISO code	EIA code	Notation in this manual
Program start	LF	CR	;

> **NOTE**
> If one file contains multiple programs, the EOB code for label skip operation must not appear before a second or subsequent program number.

- Comment section

Any information enclosed by the control-out and control-in codes is regarded as a comment.
The user can enter a header, comments, directions to the operator, etc. in a comment section.

Table 13.1 (c) Codes of a control-in and a control-out

Name	ISO code	EIA code	Notation in this manual	Meaning
Control-out	(	2-4-5	(	Start of comment section
Control-in	)	2-4-7	)	End of comment section

When a program is read into memory for memory operation, comment sections, if any, are not ignored but are also read into memory. Note, however, that codes other than those listed in the code table in Appendix A are ignored, and thus are not read into memory.
When data in memory is output on external I/O device(See III-8), the comment sections are also output.
When a program is displayed on the screen, its comment sections are also displayed. However, those codes that were ignored when read into memory are not output or displayed.
During memory operation or DNC operation, all comment sections are ignored.
The TV check function can be used for a comment section by setting parameter CTV (bit 1 of No. 0100).

⚠ **CAUTION**
If a long comment section appears in the middle of a program section, a move along an axis may be suspended for a long time because of such a comment section. So a comment section should be placed where movement suspension may occur or no movement is involved.

NOTE
1 If only a control-in code is read with no matching control-out code, the read control-in code is ignored.
2 The following codes cannot be used in the comment section:
- EOB
- % (ER for EIA)

- Program code end

A program code end is to be placed at the end of a file containing NC programs.
If programs are entered using the automatic programming system, the mark need not be entered.
The mark is not displayed on the screen. However, when a file is output, the mark is automatically output at the end of the file.
If an attempt is made to execute % when M02 or M03 is not placed at the end of the program, the alarm PS5010 is occurred.

Table 13.1 (d) Code of a program code end

Name	ISO code	EIA code	Notation in this manual
Program code end	%	ER	%

13.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See II-13.1 for program components other than program sections.

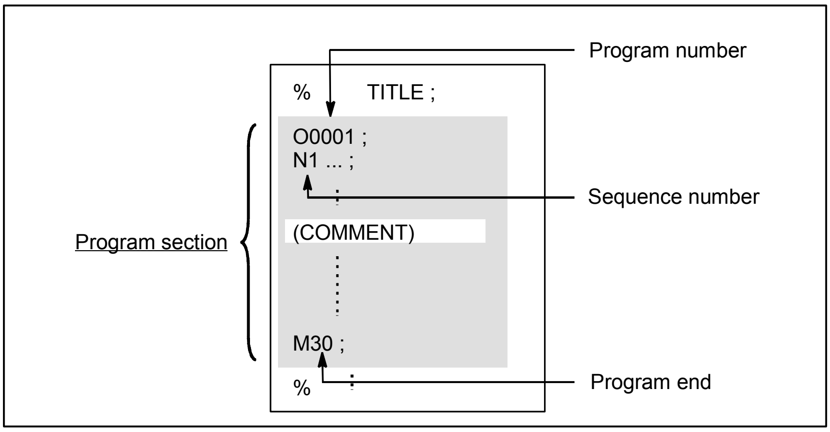


Fig. 13.2 (a) Program configuration

- Program number

A program number consisting of address O followed by a four-digit number is assigned to each program at the beginning registered in memory to identify the program. When the 8-digit number function is selected, the program number consists of eight digits (see II-13.4).
In ISO code, the colon (:) can be used instead of O.
When no program number is specified at the start of a program, the sequence number (N....) at the start of the program is regarded as its program number. If a five-digit sequence number is used, the lower four digits are registered as a program number. If the lower four digits are all 0, the program number registered immediately before added to 1 is registered as a program number. Note, however, that N0 cannot be used for a program number.
If there is no program number or sequence number at the start of a program, a program number must be specified using the MDI panel when the program is stored in memory (See III-8.4 or III-9.1)

> **NOTE**
> Program numbers 8000 to 9999 may be used by machine tool builders, and the user may not be able to use these numbers.

- File name

A file name can be assigned instead of a program number.
When coding a file name, be sure to place the file name enclosed in "<" and ">" at the beginning of a program.

Example) % ;
<PARTS_1> ;
N1 ...
:
M30 ;
%

> **NOTE**
> A file name can be coded:
> - At the beginning of a program
> - Immediately after M98, G65, G66, G66.1, M96, G72.1, or G72.2
>
> Do not code a file name in other than the above.

- Sequence number and block

A program consists of several commands. One command unit is called a block. One block is separated from another with an EOB of end of block code.

Table 13.2 (e) EOB code

Name	ISO code	EIA code	Notation in this manual
End of block (EOB)	LF	CR	;

At the head of a block, a sequence number consisting of address N followed by a number not longer than eight digits (1 to 99999999) can be placed. Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps (for example, when a new tool is used by tool replacement, and machining proceeds to a new surface with table indexing.)

```
N300X200.0Z300.0 ;    A sequence number is underlined.
```

Fig. 13.2 (b) Sequence number and block (example)

> **NOTE**
> N0 must not be used for the reason of file compatibility with other CNC systems.
> Program number 0 cannot be used. So 0 must not be used for a sequence number regarded as a program number.

- TV check (Vertical parity check)

A parity check is made for each block of input data. If the number of characters in one block (starting with the code immediately after an EOB and ending with the next EOB) is odd, a P/S alarm (No.002) is output.

No TV check is made only for those parts that are skipped by the label skip function. Bit 1 (CTV) of parameter No. 0100 is used to specify whether comments enclosed in parentheses are counted as characters during TV check. The TV check function can be enabled or disabled by setting on the MDI unit (See III-12.3.1.).

- Block configuration (word and address)

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus sign (-) may be prefixed to a number.)

For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address.

Word = Address + number (Example : X-1000)

Table 13.2 (f) indicates the usable addresses and their meanings.

The same address may have different meanings, depending on the preparatory function specification.

Table 13.2 (g) Major functions and addresses

Function	Address	Meaning
Program number	O(*)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)
Dimension word	X, Y, Z, U, V, W, A, B, C	Coordinate axis move command
	I, J, K	Coordinate of the arc center
	R	Arc radius
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
	B	Table indexing, etc.
Program number designation	P	Subprogram number
Number of repetitions	P, L	Number of subprogram repetitions
Parameter	P, Q	Canned cycle parameter

M

Offset number	D, H	Offset number
Dwell	P, X	Dwell time

T

Dwell	P, X, U	Dwell time

NOTE
(*) In ISO code, the colon (:) can also be used as the address of a program number.

N_	G_	X_ Y_	F_	S_	T_	M_	;
Sequence number	Preparatory function	Dimension word	Feed-function	Spindle speed function	Tool function	Auxiliary function	

Fig. 13.2 (c) 1 block (example)

- Major addresses and ranges of command values

Major addresses and the ranges of values specified for the addresses are shown below. Note that these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example, the CNC allows a tool to traverse up to about 100 m (in millimeter input) along the X axis.
However, an actual stroke along the X axis may be limited to 2 m for a specific machine tool.
Similarly, the CNC may be able to control a cutting feedrate of up to 240 m/min, but the machine tool may not allow more than 3 m/min. When developing a program, the user should carefully read the manuals of the machine tool as well as this manual to be familiar with the restrictions on programming.

Table 13.2 (d) Major addresses and ranges of command values

Function		Address	Input in mm	Input in inch
Program number		O (*1)	1 to 99999999	1 to 99999999
Sequence number		N	1 to 99999999	1 to 99999999
Preparatory function		G	0 to 99.9	0 to 99.9
Dimension word	Increment system IS-A	X,Y,Z,U,V, W,A,B,C,I,J ,K,R (*2)	±999999.99 mm ±999999.99 deg.	±99999.999 inch *3 ±999999.99 deg.
	Increment system IS-B		±999999.999 mm ±999999.999 deg.	±99999.9999 inch *3 ±999999.999 deg.
	Increment system IS-C		±99999.9999 mm ±99999.9999 deg.	±9999.99999 inch *3 ±99999.9999 deg.
	Increment system IS-D		±9999.99999 mm ±9999.99999 deg.	±999.999999 inch *3 ±9999.99999 deg.
	Increment system IS-E		±999.999999 mm ±999.999999 deg.	±99.9999999 inch *3 ±999.999999 deg.
Feed per minute	Increment system IS-A	F	0.01 to 999000.00mm/min	0.001 to 96000.000inch/min
	Increment system IS-B		0.001 to 999000.000mm/min	0.0001 to 9600.0000inch/min
	Increment system IS-C		0.0001 to 99999.9999mm/min	0.00001 to 4000.00000inch/min
	Increment system IS-D		0.00001 to 9999.99999mm/min	0.000001 to 400.000000inch/min
	Increment system IS-E		0.000001 to 999.999999mm/min	0.0000001 to 40.0000000inch/min
Feed per revolution		F	0.001 to 50000mm/rev	0.0001 to 50.0000inch/rev
Spindle speed function		S (*4)	0 to 99999999	0 to 99999999
Tool function		T (*4)	0 to 99999999	0 to 99999999
Auxiliary function		M (*4)	0 to 99999999	0 to 99999999
		B (*4)	0 to 99999999	0 to 99999999
Offset number (M series only)		H, D	0 to 999	0 to 999

Function		Address	Input in mm	Input in inch
Dwell	Increment system IS-A	X, U (T series only)	0 to 999999.99 sec	0 to 999999.99 sec
	Increment system IS-B		0 to 99999.999 sec	0 to 99999.999 sec
	Increment system IS-C		0 to 9999.9999 sec	0 to 9999.9999 sec
	Increment system IS-D		0 to 999.99999 sec	0 to 999.99999 sec
	Increment system IS-E		0 to 99.999999 sec	0 to 99.999999 sec
Dwell		P	1 to 99999999	1 to 99999999
Designation of a program number		P	1 to 99999999	1 to 99999999
Number of subprogram repetitions		L	1 to 99999999	1 to 99999999
		P	0 to 9999	0 to 9999

*1 In ISO code, the colon (:) can also be used as the address of a program number.

*2 When address I, J, K, or R is used to specify the radius for circular interpolation, the specifiable range is as follows:

Increment system	Input in mm	Input in inch
IS-A	±999999999.99 mm	±99999999.999 inch
IS-B	±999999999.999 mm	±99999999.9999 inch
IS-C	±99999999.9999 mm	±9999999.99999 inch
IS-D	±9999999.99999 mm	±999999.999999 inch
IS-E	±999999.999999 mm	±99999.9999999 inch

*3 For inch input/millimeter machines, the maximum specifiable range of dimension words is as follows:

Increment system	Maximum specifiable range
IS-A	±39370.078 inch
IS-B	±39370.0787 inch
IS-C	±3937.00787 inch
IS-D	±393.700787 inch
IS-E	±39.3700787 inch

*4 The maximum value of addresses M, S, T, and B is 99999999. Note that, however, values longer than the permissible number of digits set in parameter No. 3030 to 3033 cannot be specified. The values and uses for some codes are limited by parameter setting. (For example, some M codes are not buffered.) For details, refer to the parameter manual.

- Optional block skip

When a slash followed by a number (/n (n=1 to 9)) is specified at the head of a block, and optional block skip signals BDT1 to BDT9 are set to 1 during automatic operation, the information (/n to the end of the block (EOB)) contained in the block for which /n corresponding to signal BDTn is specified is ignored.

Example 1)
/2 N123 X100.0 Y200.0 ;

Example 2)
//3 N123 X100.0 Y200.0 ; → Incorrect
/1 /3 N123 X100.0 Y200.0 ; → Correct

Input signal and program code

Input signal	Start code to be ignored
BDT1	/ or /1(NOTE)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

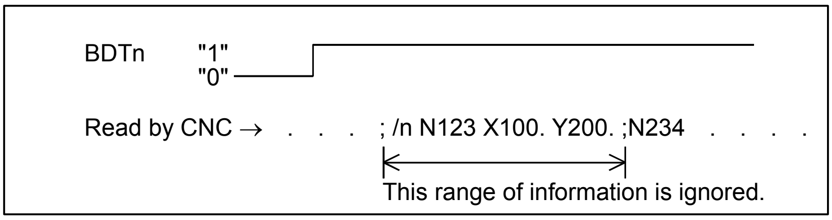
NOTE

1 Number 1 for /1 can be omitted. However, when two or more optional block skips are specified for one block, number 1 for /1 cannot be omitted.

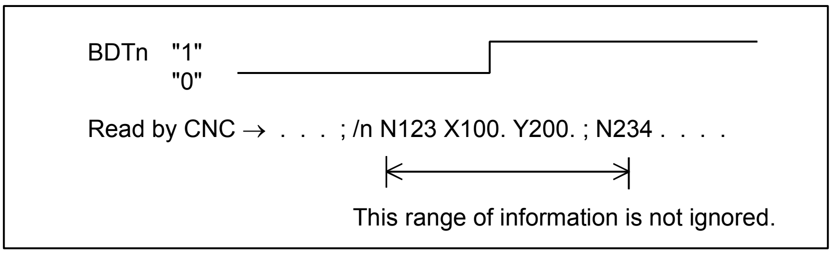
2 Depending on the machine tool, all optional block skip signals (1 to 9) may not be usable. Refer to manuals of the machine tool builder to find which switches are usable.

The following shows the relationship between the timing at which optional block skip signals BDT1 to BDT9 are set to 1 and the range of information to be ignored.

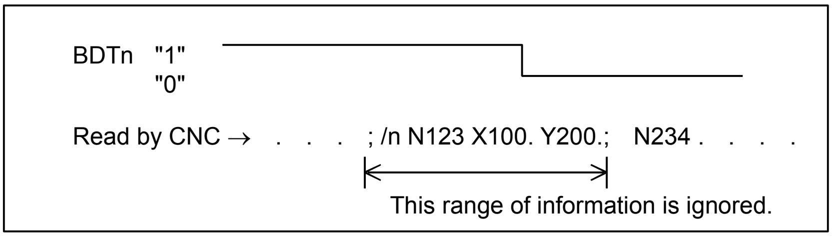
1. When the signal BDTn is set to 1 before the CNC starts reading a block that contains /n, the block is ignored.



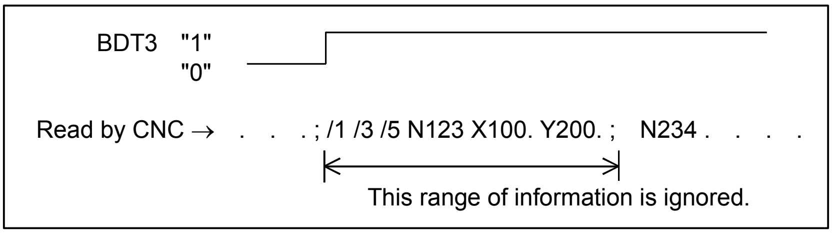
2. When the signal BDTn is set to 1 while the CNC is reading a block that contains /n, the block is not ignored.



3. When the signal BDTn is set to 0 while the CNC is reading a block that contains /n, the block is ignored.



4. Two or more optional block skips can be specified in one block. When the signal corresponding to any of the specified skips is set to 1, the block is ignored.



NOTE

1 This function is not used when a program is registered in memory. A block containing / is registered in memory regardless of the statuses of optional block skip signals. When a program in memory is also output regardless of the statuses of optional block skip signals.
In addition, the optional block skip function is enabled during a search for a sequence number.

2 Position of a slash
A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

3 TV and TH check
When the optional block skip signal is set to 1, TH and TV checks are made for the skipped portions in the same way as when the optional block skip signal is set to 0.

- Program end

The end of a program is indicated by programming one of the following codes at the end of the program:

Table 13.2 (h) Code of a program end

Code	Meaning usage
M02	For main program
M03	
M99	For subprogram

If one of the program end codes is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. When the subprogram end code is executed, control returns to the program that called the subprogram.

> ⚠ **CAUTION**
> A block containing an optional block skip code such as /M02 ; , /M30 ; , or /M99 ; is not regarded as the end of a program. (see "Optional block skip".)

13.3 SUBPROGRAM (M98, M99)

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.
A subprogram can be called from the main program.
A called subprogram can also call another subprogram.

Format

- Subprogram configuration

One subprogram

Oxxxx ;	Subprogram number or subprogram file name (or the colon (:) optionally in the case of ISO)
:	
:	
M99;	Program end

M99 need not constitute a separate block as indicated below.

Example) X100.0Y100.0M99;

- Subprogram call

- **When a subprogram with a 4-digit or shorter program number is called**

M98 P<u>xxxx</u> <u>xxxx</u> ;

Subprogram number
Number of times the subprogram is called repeatedly

- **When a subprogram with a 5-digit or longer program number is called**

M98 P<u>xxxxxxxx</u> <u>Lxxxxxxxx</u> ;

Subprogram number
Number of times the subprogram is called repeatedly

- **When a subprogram is called by file name**

M98 <<u>xxxx</u>> <u>Lxxxxxxxx</u> ;

Subprogram file name
Number of times the subprogram is called repeatedly

- Called program and folders to be searched

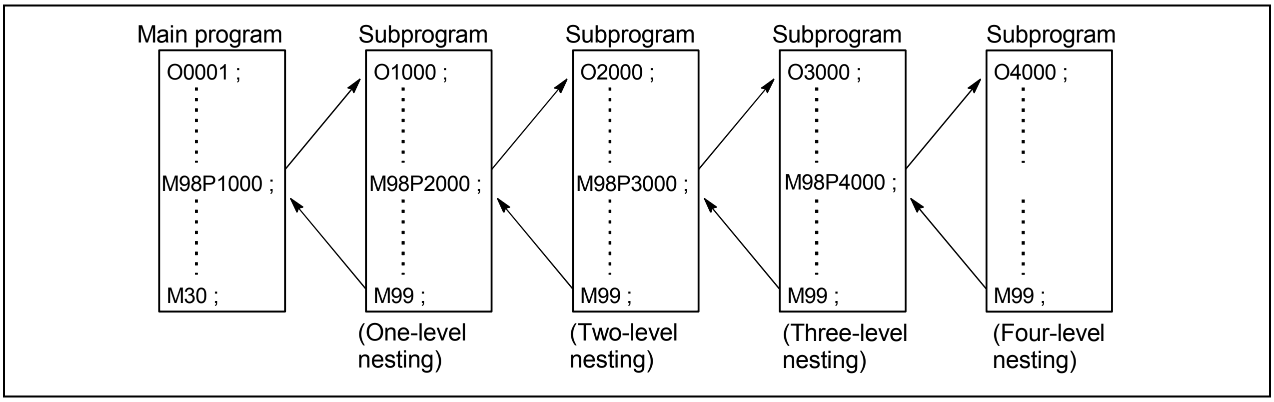
The order in which folders are searched depends on the method of calling a subprogram. Folders are searched in sequence and the program found first is called. For details, see the "Managing Programs" chapter.

NOTE

1. When a subprogram with a subprogram number shorter than 4 digits is called, the length of the subprogram number must be adjusted to 4 digits by adding 0(s) to the beginning of the program number.
   Example)
   P100100: Call subprogram No. 100 ten times.
   P50001: Call subprogram No. 1 five times.
2. If the number of times the subprogram is called repeatedly is omitted when a subprogram with a 4-digit or shorter program number is called, the subprogram is called just once.
   In this case, it is not necessary to adjust the subprogram number length to 4 digits as described in Item 1 above.
3. When calling a subprogram with a 4-digit or shorter program number, do not specify address L in the same block.
4. When calling a subprogram with a 5-digit or longer program number, do not omit the specification of the number of repeats.
5. When calling a subprogram by file name, be sure to specify the file name immediately after M98.

Explanation

When the main program calls a subprogram, it is regarded as a one-level subprogram call. Thus, subprogram calls can be nested up to four levels as shown below.



A single call command can repeatedly call a subprogram up to 99999999 times. For compatibility with automatic programming systems, in the first block, Nxxxxxxxx can be used instead of a

subprogram number that follows O (or :). A sequence number after N is registered as a subprogram number.

NOTE
1. The M98 and M99 code signal and strobe signal are not output to the machine tool.
2. If the subprogram number specified by address P cannot be found, an alarm PS0078 is output.
3. When an attempt is made to call a subprogram by file name, but the specified file cannot be found, alarm PS0310 is issued.

Example

- M98 P51002 ;

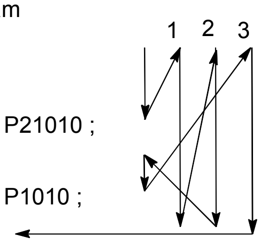
  This command specifies "Call the subprogram (number 1002) five times in succession." A subprogram call command (M98P_) can be specified in the same block as a move command.

- X1000.0 M98 P1200 ;

  This example calls the subprogram (number 1200) after an X movement.

- Execution sequence of subprograms called from a main program

Main program	Subprogram
N0010 . . . ;	O1010 . . . ;
N0020 . . . ;	N1020 . . . ;
N0030 M98 P21010 ;	N1030 . . . ;
N0040 . . . ;	N1040 . . . ;
N0050 M98 P1010 ;	N1050 . . . ;
N0060 . . . ;	N1060 . . . M99 ;



A subprogram can call another subprogram in the same way as a main pro gram calls a subprogram.

Special usage

- Specifying the sequence number for the return destination in the main program

If P is used to specify a sequence number when a subprogram is terminated, control does not return to the block after the calling block, but returns to the block with the sequence number specified by P. Note, however, that P is ignored if the main program is operating in a mode other than memory operation mode.
This method consumes a much longer time than the normal return method to return to the main program.

```
Main program                          Subprogram
N0010 . . . ;                         O0010 . . . ;
N0020 . . . ;                         N1020 . . . ;
N0030 M98 P1010 ;                     N1030 . . . ;
N0040 . . . ;                         N1040 . . . ;
N0050 . . . ;                         N1050 . . . ;
N0060 . . . ;                         N1060 . . . M99 P0060 ;
```

- Using M99 in the main program

If M99 is executed in a main program, control returns to the start of the main program. For example, M99 can be executed by placing /M99 ; at an appropriate location of the main program and setting the optional block skip function to off when executing the main program. When M99 is executed, control returns to the start of the main program, then execution is repeated starting at the head of the main program.
Execution is repeated while the optional block skip function is set to off.
If the optional block skip function is set to on, the /M99 ; block is skipped ; control is passed to the next block for continued execution.
If/M99Pn ; is specified, control returns not to the start of the main program, but to sequence number n. In this case, a longer time is required to return to sequence number n.

```
                       N0010 . . . ;
                       N0020 . . . ;
                       N0030 . . . ;
                       N0040 . . . ;
Optional block skip    N0050 . . . ;
OFF                  / N0060 . . . M99 P0030 ;   Optional block
                       N0070 . . . ;             skip ON
                       N0080 M02 ;
```

- Using a subprogram only

A subprogram can be executed just like a main program by searching for the start of the subprogram with the MDI.

(See III-10.4 for information about search operation.)

In this case, if a block containing M99 is executed, control returns to the start of the subprogram for repeated execution. If a block containing M99Pn is executed, control returns to the block with sequence number n in the subprogram for repeated execution. To terminate this program, a block containing /M02 ; or /M30 ; must be placed at an appropriate location, and the optional block switch must be set to off ; this switch is to be set to on first.

```
 N1010 . . . ;
 N1020 . . . ;
 N1030 . . . ;
/N1040 . . . M02 ;
 N1050 M99 P1020 ;      Optional block skip
                        ON
```

14 FUNCTIONS TO SIMPLIFY PROGRAMMING

This chapter explains the following items:

14.1 FIGURE COPY (G72.1, G72.2)

Machining can be repeated after moving or rotating the figure using a subprogram.

Format

- Rotational copy

Xp-Yp plane (specified by G17) :
G72.1 P_ L_ Xp_Yp_R_ ;
Zp-Xp plane (specified by G18) :
G72.1 P_ L_ Zp_Xp_R_ ;
Yp-Zp plane (specified by G19) :
G72.1 P_ L_ Yp_Zp_R _;

P :Subprogram number
L :Number of times the operation is repeated
Xp :Center of rotation on the Xp axis
(Xp : X-axis or an axis parallel to the X-axis)
Yp :Center of rotation on the Yp axis
(Yp: Y-axis or an axis parallel to the Y-axis)
Zp :Center of rotation on the Zp axis
(Zp: Z-axis or an axis parallel to the Z-axis)
R :Angular displacement
(A positive value indicates a counterclockwise angular displacement. Specify an incremental value.)
Specify a plane selection command (G17, G18, or G19) to select the plane on which the rotational copy is made.

- Linear copy

Xp-Yp plane (specified by G17) :
G72.2 P_ L_ I_ J_ ;
Zp-Xp plane (specified by G18) :
G72.2 P_ L_ K_ I_ ;
Yp-Zp plane (specified by G19) :
G72.2 P_ L_ J_ K_;
P :Subprogram number
L :Number of times the operation is repeated
I :Shift along the Xp axis
J :Shift along the Yp axis
K :Shift along the Zp axis
Specify a plane selection command (G17, G18, or G19) to select the plane on which the linear copy is made.

Explanation

- First block of the subprogram

Always specify a move command in the first block of a subprogram that performs a rotational or linear copy. If the first block contains only the program number such as O1234; and does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1, 2, 3, ...) copying.

Specify the first move command in the absolute mode.

(Example of an incorrect program)

```
O1234 ;
G00 G90 X100.0 Y200.0 ;
. . . . . ;
. . . . . ;
M99 ;
```

(Example of a correct program)

```
O1000 G00 G90 X100.0 Y200.0 ;
. . . . . ;
. . . . . ;
M99 ;
```

- Combination of rotational and linear copying

The linear copy command can be specified in a subprogram for a rotational copy. Also, the rotational copy command can be specified in a subprogram for a linear copy.

- Subprogram call

In a subprogram for rotational or linear copying, M98 for calling another subprogram or G65 for calling a macro can be specified.

- Specifying the center of rotation

The center of rotation specified with G72.1 is processed as an absolute position even in the incremental mode.

- Specifying address

In a block with G72.1, addresses other than P, L, Xp, Yp, Zp, or R are ignored. The subprogram number (P), coordinates of the center of rotation (Xp, Yp, Zp), and angular displacement (R) must be specified.

In a block with G72.2, addresses other than P, L, I, J, or K are ignored. The subprogram number (P) and shift (I, J, K) must be specified.

- Address P

If the subprogram number specified with P is not found, alarm PS0078 occurs. If P is not specified, alarm PS0076 occurs.

- Address L

If L is omitted, the repetition count is assumed to be 1 and the subprogram is called only once.

- Increment in angular displacement or shift

In a block with G72.1, an increment in angular displacement is specified with address R. The angular displacement of the figure made by the n-th rotation is calculated as follows : R × (n - 1).
In a block with G72.2, an increment in shift is specified with addresses I, J, and K. The shift of the figure made by the n-th movement is calculated as follows : (Programmed shift) x (n - 1).

- Nesting level of a subprogram

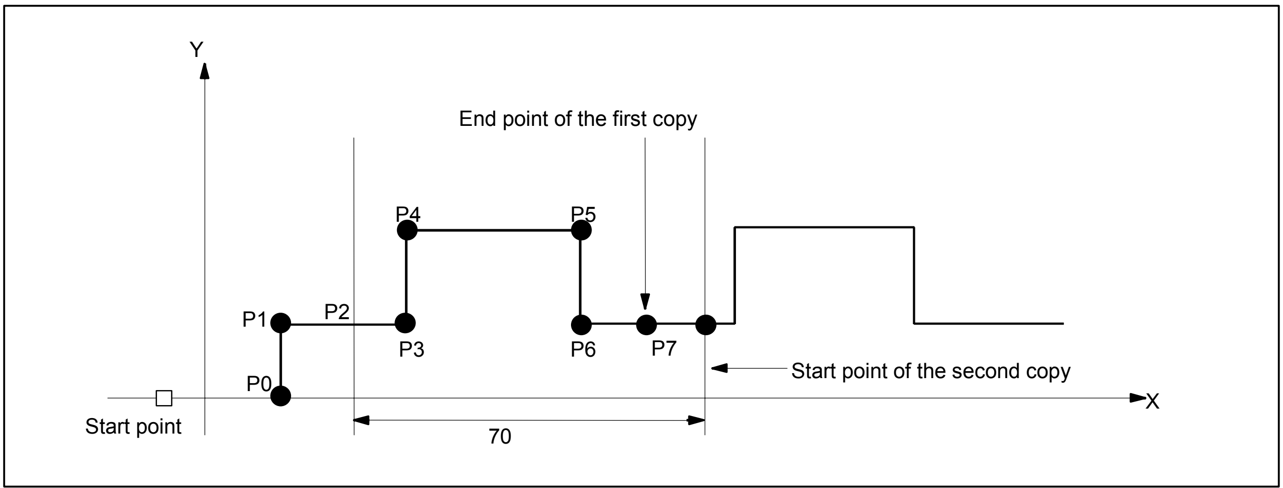
If a subprogram is called by G72.1 or G72.2, the nesting level is increased by one in the same manner as when M98 is specified.

- Block end position

The coordinates of a figure moved rotationally or linearly (block end position) can be read from #5001 and subsequent system variables of the custom macro of rotational or linear copy.

- Disagreement between end point and start point

If the end point of the figure made by the n-th copy does not agree with the start point of the figure to be made by the next (n + 1) copy, the figure is moved from the end point to the start point, then copying is started. (Generally, this disagreement occurs if an incorrect angular displacement or shift is specified.)




```
Main program
  O1000 ;
  N10 G92 X-20.0 Y0 ;
  N20 G00 G90 X0 Y0 ;
  N30 G01 G17 G41 X20. Y0 D01 F10 ;   (P0)
  N40 Y20. ;                          (P1)
  N50 X30. ;                          (P2)
  N60 G72.2 P2000 L3 I90. J0 ;
                        Although a shift of 70 mm was
                        required, I90.0 was specified instead
                        of I70.0. Since an incorrect shift was
                        specified, the end point of the figure
                        made by the n-th copy disagrees
                        with the start point of the figure to be
                        made by the next (n + 1) copy.

Subprogram
  O2000 G90 G01 X40. ;                (P3)
  N100 Y40. ;                         (P4)
  N200 G01 X80. ;                     (P5)
  N300 G01 Y20. ;                     (P6)
  N400 X100. ;                        (P7)
  N500 M99;
```

Limitation

- Specifying two or more commands to copy a figure

G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, alarm PS0160 will occur). G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, alarm PS0161 will occur).

- Commands that must not be specified

Within a program that performs a rotational or linear copy, the following must not be specified:

- Command for changing the selected plane (G17 to G19)
- Command for specifying polar coordinates
- Reference position return command
- Coordinate system rotation, scaling, programmable mirror image

The command for rotational or linear copying can be specified after a command for coordinate system rotation, scaling, or programmable mirror image is executed.

- Modes that must not be selected

The figure cannot be copied during chamfering, corner rounding, or tool offset.

- Unit system

The two axes of the plane for copying a figure must have an identical unit system.

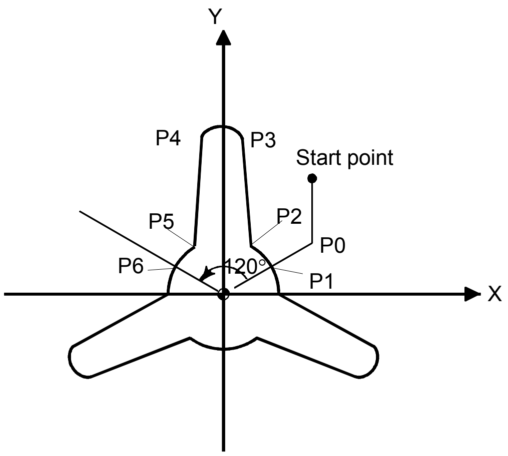
- Single block

Single-block stops are not performed in a block with G721.1 or G72.2.

- Specifying cutter compensation and the workpiece coordinate system

In a subprogram for copying a figure, the G code for cutter compensation B or C or compensation amount (H or D code) cannot be changed. G92 and G54 to G59 cannot be changed either. Those codes must be specified before figure copying is started.

Example
- Rotational copy



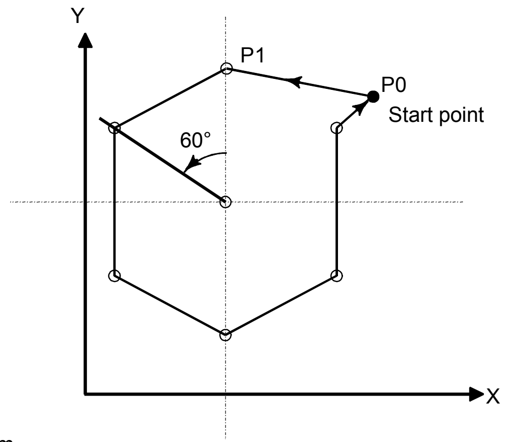
Main program

```
O1000 ;
N10 G92 X40.0 Y50.0 ;
N20 G00 G90 X_ Y_ ;                    (P0)
N30 G01 G17 G41 X_ Y_ D01 F10 ;        (P1)
N40 G72.1 P2000 L3 X0 Y0 R120.0 ;
N50 G40 G01 X_ Y_ I_ J_ ;              (P0)
N60 G00 X40.0 Y50.0 ;
N70 M30 ;
```

Sub program

```
O2000 G03 X_ Y_ R30.0 ;                (P2)
N100 G01 X_ Y_ ;                       (P3)
N200 G03 X_ Y_ R10.0 ;                 (P4)
N300 G01 X_ Y_ ;                       (P5)
N400 G03 X_ Y_ R30.0 ;                 (P6)
N500 M99;
```

- Rotational copy (spot boring)



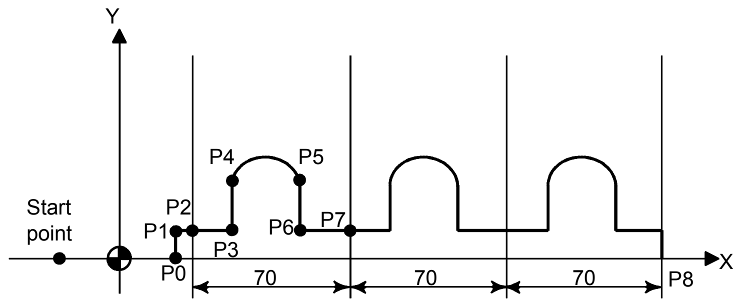
Main program

```
O3000 ;
N10 G92 G17 X80.0 Y50.0 ;                (P0)
N20 G72.1 P4000 L6 X0 Y0 R60.0 ;
N30 G80 G00 X80.0 Y50.0 ;                (P0)
N40 M30 ;
```

Subprogram

```
O4000 N100 G90 G81 X_ Y_ R_ Z_ F_ ;      (P1)
N200 M99 ;
```

- Linear copy



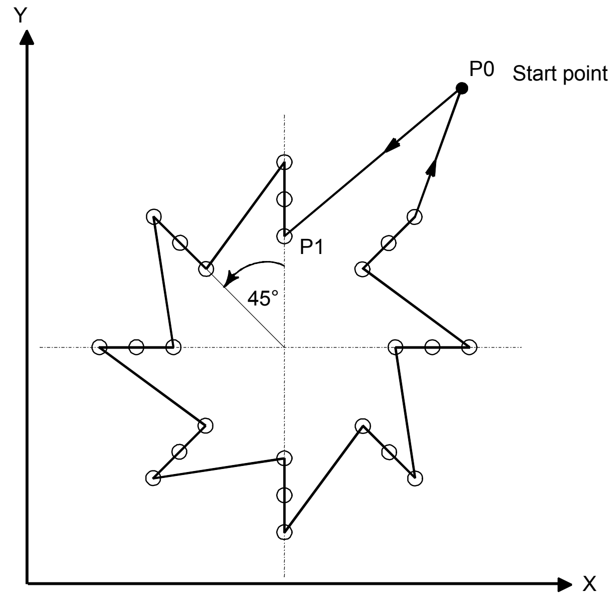
Main program

```
O1000 ;
N10 G92 X-20.0 Y0 ;
N20 G00 G90 X0 Y0 ;
N30 G01 G17 G41 X20. Y0 D01 F10 ;   (P0)
N40 Y20. ;                          (P1)
N50 X30. ;                          (P2)
N60 G72.2 P2000 L3 I70.0 J0 ;
N70 X_ Y_ ;                         (P8)
N80 X0 ;
N90 G00 G40 X-20.0 Y0 ;
N100 M30 ;
```

Subprogram

```
O2000 G90 G01 X_ ;                  (P3)
N100 Y_ ;                           (P4)
N200 G02 X_ I_ ;                    (P5)
N300 G01 Y_ ;                       (P6)
N400 X_ ;                           (P7)
N500 M99 ;
```

- Combination of rotational copying and linear copying (bolt hole circle)



Main program

```
O1000 ;
N10 G92 G17 X100.0 Y80.0 ;          (P0)
N20 G72.1 P2000 X0 Y0 L8 R45.0 ;
N30 G80 G00 X100.0 Y80.0 ;          (P0)
N40 M30 ;
```

Subprogram (rotational copy)

```
O2000 N100 G72.2 P3000 I0 J_ L3 ;
N200 M99 ;
```

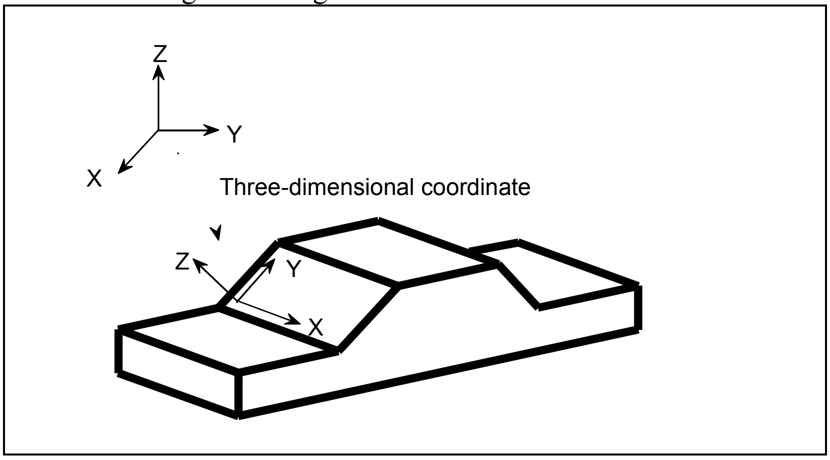
Subprogram (linear copy)

```
O3000 N110 G90 G81 X_ Y_ R_ Z_ F_ ;     (P1)
N210 M99 ;
```

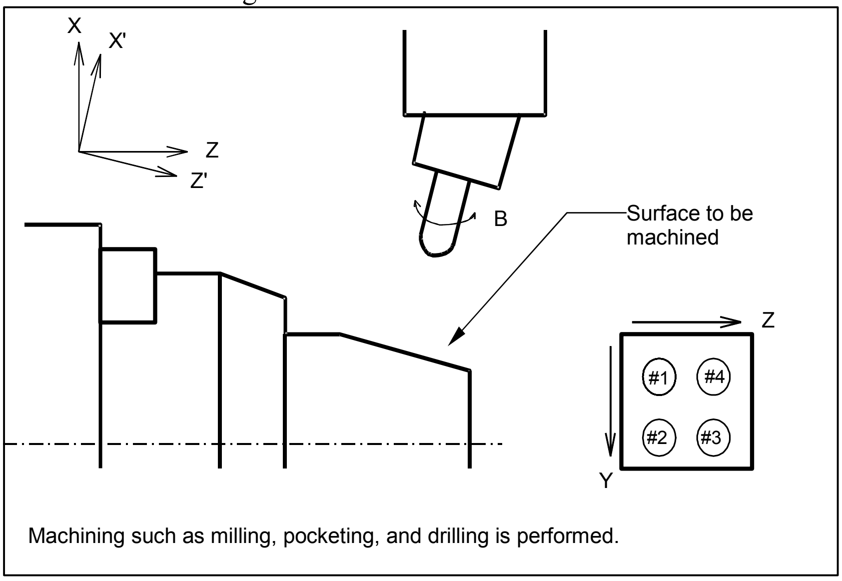
14.2 THREE-DIMENSIONAL COORDINATE CONVERSION

Coordinate conversion about an axis can be carried out if the center of rotation, direction of the axis of rotation, and angular displacement are specified. This function is very useful in three-dimensional machining by a die-sinking machine or similar machine. For example, if a program specifying machining on the XY plane is converted by the three-dimensional coordinate conversion function, the identical machining can be executed on a desired plane in three-dimensional space.

- For milling machining



- For lathe cutting



Format

M

G68 Xp$\underline{x_1}$ Yp$\underline{y_1}$ Zp$\underline{z_1}$ I$\underline{i_1}$ J$\underline{j_1}$ K$\underline{k_1}$ R$\underline{\alpha}$;	Starting three-dimensional coordinate conversion
: **:**	Three-dimensional coordinate conversion mode
G69 ;	Canceling three-dimensional coordinate conversion

Xp, Yp, Zp : Center of rotation (absolute coordinates) on the X, Y, and Z axis or parallel axes
I, J, K : Direction of the axis of rotation
R : Angular displacement

T

G68.1 Xp$\underline{x_1}$ Yp$\underline{y_1}$ Zp$\underline{z_1}$ I$\underline{i_1}$ J$\underline{j_1}$ K$\underline{k_1}$ R$\underline{\alpha}$;	Starting three-dimensional coordinate conversion
: **:**	Three-dimensional coordinate conversion mode
G69.1 ;	Canceling three-dimensional coordinate conversion

Xp, Yp, Zp : Center of rotation (absolute coordinates) on the X, Y, and Z axis or parallel axes
I, J, K : Direction of the axis of rotation
R : Angular displacement

NOTE
The G code of this function is hereinafter described using the format (G68/G69) for the machining center system in this section.

Explanation

- Command for three-dimensional coordinate conversion (program coordinate system)

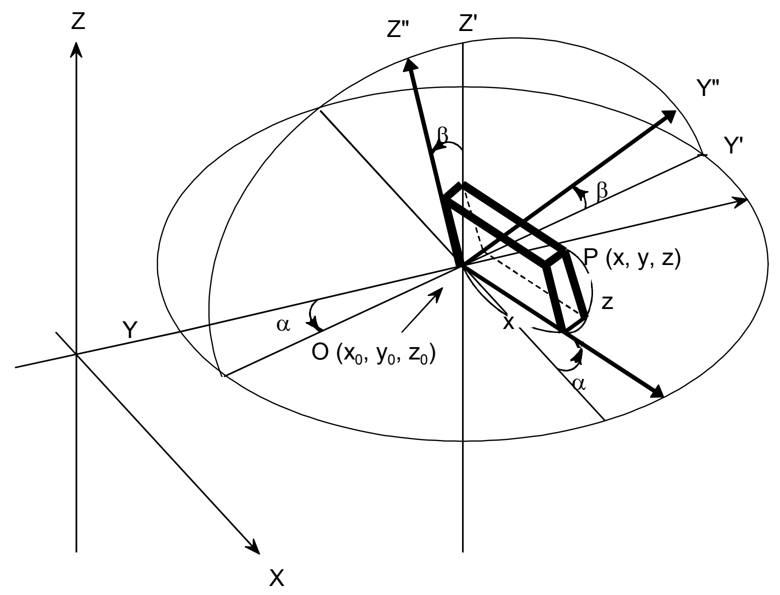
N1 G68 Xp $\underline{x_1}$ Yp $\underline{y_1}$ Zp $\underline{z_1}$ I $\underline{i_1}$ J $\underline{j_1}$ K $\underline{k_1}$ R $\underline{\alpha}$;
N2 G68 Xp $\underline{x_2}$ Yp $\underline{y_2}$ Zp $\underline{z_2}$ I $\underline{i_2}$ J $\underline{j_2}$ K $\underline{k_2}$ R $\underline{\beta}$;
N3
:
Nn G69 ;

Three-dimensional coordinate conversion can be executed twice.
In the N1 block, specify the center, direction of the axis of rotation, and angular displacement of the first rotation.

When this block is executed, the center of the original coordinate system is shifted to (x_1, y_1, z_1), then rotated around the vector (i_1, j_1, k_1) by angular displacement α. The new coordinate system is called X'Y'Z'. In the N2 block, specify the center, direction of the axis of rotation, and angular displacement of the second rotation. In the N2 block, specify coordinates and the angle with the coordinate system formed after the N1 block in Xp, Yp, Zp, I, J, K, and R. When the N2 block is executed, the X'Y'Z' coordinate system is shifted to (x_2, y_2, z_2), then rotated around the vector (i_2, j_2, k_2) by angular displacement β. The newest coordinate system is called X"Y"Z". In the subsequent N3 block, coordinates in the X"Y"Z" coordinate system are specified with Xp, Yp, and Zp. The X"Y"Z" coordinate system is called the program coordinate system.

If (Xp, Yp, Zp) is not specified in the N2 block, (Xp, Yp, Zp) in the N1 block is assumed to be the center of the second rotation (the N1 and N2 blocks have a common center of rotation). If the coordinate system is to be rotated only once, the N2 block need not be specified.

Example) G68 Xx_0 Yy_0 Zz_0 I0 J0 K1 Rα ;
G68 I1 J0 K0 Rβ ;



X, Y, Z	: Workpiece coordinate system
X', Y', Z'	: Coordinate system formed after the first conversion
X", Y", Z"	: Coordinate system formed after the second conversion
α	: Angular displacement of the first rotation
β	: Angular displacement of the second rotation
O (x_0, y_0, z_0)	: Center of rotation
P (x, y, z)	: Coordinates in the X"Y"Z" coordinate system (program coordinate system)

- Format error

If one of the following format errors is detected, alarm PS5044 occurs:

1. When I, J, or K is not specified in a block with G68
   (a parameter of coordinate system rotation is not specified)
2. When I, J, and K are all set to 0 in a block with G68
3. When R is not specified in a block with G68

- Center of rotation

Specify absolute coordinates with Xp, Yp, and Zp in the G68 block.

- Equation for three-dimensional coordinate conversion

The following equation shows the general relationship between (x, y, z) in the program coordinate system and (X, Y, Z) in the original coordinate system (workpiece coordinate system).

$$\begin{pmatrix} X \\ Y \\ Z \end{pmatrix} = (M_1)\begin{pmatrix} x \\ y \\ z \end{pmatrix} + \begin{pmatrix} x_1 \\ y_1 \\ z_1 \end{pmatrix}$$

When conversion is carried out twice, the relationship is expressed as follows:

$$\begin{pmatrix} X \\ Y \\ Z \end{pmatrix} = (M_1)(M_2)\begin{pmatrix} x \\ y \\ z \end{pmatrix} + (M_1)\begin{pmatrix} x_2 \\ y_2 \\ z_2 \end{pmatrix} + \begin{pmatrix} x_1 \\ y_1 \\ z_1 \end{pmatrix}$$

X, Y, Z : Coordinates in the original coordinate system (workpiece coordinate system)
x, y, z : Programmed value (coordinates in the program coordinate system)
x_1, y_1, z_1 : Center of rotation of the first conversion
x_2, y_2, z_2 : Center of rotation of the second conversion (coordinates in the coordinate system formed after the first conversion)
M_1 : First conversion matrix
M_2 : Second conversion matrix

M_1 and M_2 are conversion matrices determined by an angular displacement and rotation axis. Generally, the matrices are expressed as shown below.

$$\begin{pmatrix} n_1^2 + (1-n_1^2)\cos\theta & n_1 n_2(1-\cos\theta) - n_3\sin\theta & n_1 n_3(1-\cos\theta) + n_2\sin\theta \\ n_1 n_2(1-\cos\theta) + n_3\sin\theta & n_2^2 + (1-n_2^2)\cos\theta & n_2 n_3(1-\cos\theta) - n_1\sin\theta \\ n_1 n_3(1-\cos\theta) - n_2\sin\theta & n_2 n_3(1-\cos\theta) + n_1\sin\theta & n_3^2 + (1-n_3^2)\cos\theta \end{pmatrix}$$

n_1 : Cosine of the angle made by the rotation axis and X-axis i/p
n_2 : Cosine of the angle made by the rotation axis and Y-axis j/p
n_3 : Cosine of the angle made by the rotation axis and Z-axis k/p
θ : Angular displacement
Value p is obtained by the following:

$$p = \sqrt{i^2 + j^2 + k^2}$$

Conversion matrices for rotation on two-dimensional planes are shown below:

(1) Coordinate conversion on the XY plane

$$M = \begin{pmatrix} \cos\theta & -\sin\theta & 0 \\ \sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{pmatrix}$$

(2) Coordinate conversion on the YZ plane

$$M = \begin{pmatrix} 1 & 0 & 0 \\ 0 & \cos\theta & -\sin\theta \\ 0 & \sin\theta & \cos\theta \end{pmatrix}$$

(3) Coordinate conversion on the ZX plane

$$M = \begin{pmatrix} \cos\theta & 0 & \sin\theta \\ 0 & 1 & 0 \\ -\sin\theta & 0 & \cos\theta \end{pmatrix}$$

- Three basic axes and their parallel axes

Three-dimensional coordinate conversion can be applied to a desired combination of three axes selected out of the basic three axes (X, Y, Z) and their parallel axes. The three-dimensional coordinate system subjected to three-dimensional coordinate conversion is determined by axis addresses specified in the G68 block. If Xp, Yp, or Zp is not specified, X, Y, or Z of the basic three axes is assumed. However, if the basic three axes are not specified in parameter 1022, alarm PS0048 occurs.

In a single G68 block, both a basic axis and a parallel axis cannot be specified.

If this is attempted, alarm PS0047 occurs.

(Example)

When U-axis, V-axis, and W-axis are parallel to the X-axis, Y-axis, and Z-axis respectively

G68 X_ I_ J_ K_ R_ ;	XYZ coordinate system
G68 U_V_ Z_ I_ J_ K_ R_ ;	UVZ coordinate system
G68 W_ I_ J_ K_ R_ ;	XYW coordinate system

- Specifying the second conversion

Three-dimensional coordinate conversion can be executed twice. The center of rotation of the second conversion must be specified with the axis addresses specified for the first conversion. If the axis addresses of the second conversion are different from the axis addresses of the first conversion, the different axis addresses are ignored. An attempt to execute three-dimensional coordinate conversion three or more times causes alarm PS5043.

- Angular displacement R

A positive angular displacement R indicates a clockwise rotation along the axis of rotation. Specify angular displacement R in 0.001 degrees within the range of -360000 to 360000.

- G codes that can be specified

The following G codes can be specified in the three-dimensional coordinate conversion mode:

G00 Positioning
G01 Linear interpolation
G02 Circular interpolation (clockwise)
G03 Circular interpolation (counterclockwise)
G04 Dwell
G10 Data setting
G17 Plane selection (XY)
G18 Plane selection (ZX)
G19 Plane selection (YZ)
G28 Reference position return
G29 Movement from the reference position
G30 Second, third, or fourth reference position return
G53 Selecting the machine coordinate system
G65 Custom macro call
G66 Custom macro modal call
G67 Custom macro modal call calling
G40 Canceling cutter or tool nose radius compensation
G41 Cutter or tool nose radius compensation to the left
G42 Cutter or tool nose radius compensation to the right
G73, G74, G76, G80 to G89 Canned cycle for drilling

M

G43 Increasing tool length compensation
G44 Decreasing tool length compensation
G45 Increasing the tool offset
G46 Decreasing the tool offset
G47 Doubling the tool offset
G48 Halving the tool offset
G49 Canceling tool length compensation
G50.1 Canceling programmable mirror image
G51.1 Programmable mirror image
G90 Absolute programming
G91 Incremental programming
G94 Feed per minute
G95 Feed per revolution
G98 Canned cycle (return to the initial level)
G99 Canned cycle (return to the level of point R)

T

G90 Absolute programming (when G code system B or C is used.)
G91 Incremental programming (when G code system B or C is used.)
G94 Feed per minute (when G code system B or C is used.)
G95 Feed per revolution(when G code system B or C is used.)
G98 Canned cycle (return to the initial level) (when G code system B or C is used.)
G99 Canned cycle (return to the level of point R) (when G code system B or C is used.)

- Rapid traverse rate in drilling of a canned cycle for drilling

In three-dimensional coordinate conversion mode, rapid traverse rate in drilling by a canned cycle for drilling equals the cutting feedrate specified in parameter No. 5412. If the parameter is set to 0, the rapid traverse rate equals the maximum cutting feedrate.

- Compensation functions

M

If tool length compensation, cutter or tool nose radius compensation, or tool offset is specified with three-dimensional coordinate conversion, compensation is performed first, followed by three-dimensional coordinate conversion.

T

If cutter or tool nose radius compensation is specified with three-dimensional coordinate conversion, compensation is performed first, followed by three-dimensional coordinate conversion.

- Relationship between three-dimensional and two-dimensional coordinate conversion

Three-dimensional and two-dimensional coordinate conversion use identical G codes (G68 and G69). A G code specified with I, J, and K is processed as the command for three-dimensional coordinate conversion. A G code not specified with I, J, and K is processed as the command for two-dimensional coordinate conversion.

- Custom macro system variables

Coordinates on the workpiece coordinate system are assigned to system variables #5041 to #5048 (current position on each axis).

- Reset

If a reset occurs during three-dimensional coordinate conversion mode, the mode is canceled and the continuous-state G code is changed to G69.

T

The parameter D3R (No. 5400#2) determines whether just the G69.1 code is used to cancel the three-dimensional coordinate conversion mode (G68.1). When this setting is selected, a CNC reset by a reset operation or by an input signal from the PMC will not cancel the three-dimensional coordinate conversion mode.

- Absolute position display

M

The absolute coordinates based on the program or workpiece coordinate system can be displayed in the three-dimensional coordinate conversion mode. Specify a desired coordinate system in the parameter DAK (No. 3106#6).

- Three-dimensional rigid tapping

By specifying the rigid tapping command in three-dimensional coordinate conversion mode, tapping can be executed in the direction of the angle programmed by the three-dimensional coordinate conversion command.
In three-dimensional coordinate conversion mode, "Position Error Z", displayed on the spindle adjustment screen, is taken from the longitudinal tapping axis after three-dimensional conversion.
Positioning in three-dimensional coordinate conversion mode must be linear interpolation positioning (the parameter LRP (No.1401#1) is set to 1).
Three-dimensional rigid tapping cannot be executed for an axis under simple synchronous control.

Limitation

- Manual intervention

Three-dimensional coordinate conversion does not affect the degree of manual intervention or manual handle interrupt.

- Positioning in the machine coordinate system

Three-dimensional coordinate conversion does not affect positioning in the machine coordinate system (e.g. specified with G28, G30, or G53).

- Specifying rapid traverse

Specify linear rapid traverse when three-dimensional coordinate conversion is executed. (Set the parameter LRP (No. 1401#1) to 1.)

- Block with G68 or G69

In a block with G68 or G69, other G codes must not be specified. G68 must be specified with I, J, and K.

- Position display and compensation

To display the absolute position when three-dimensional coordinate conversion is executed, set parameters DRL, DRC, DAL, DAC (No. 3104#4 to #7) to 0.

- Mirror image

M

Programmable mirror image can be specified, but external mirror image (mirror image by the mirror image signal or setting) cannot be specified. Three-dimensional coordinate conversion is carried out after the programmable mirror image function is executed.

T

External mirror image (mirror image by the mirror image signal or setting) cannot be specified.

- Three-dimensional coordinate conversion and other continuous-state commands

M

Canned cycles G41, G42, or G51.1 must be nested between G68 and G69.
(Example)

```
┌---  G68 X100. Y100. Z100. I0. J0. K1. R45. ;
¦       :
¦ ┌-  G41 D01 ;
¦ ¦     :
¦ └-  G40 ;
¦       :
└---  G69 ;
```

T

Canned cycles G41 or G42 must be nested between G68.1 and G69.1.
(Example)

```
┌---  G68.1 X100. Y100. Z100. I0. J0. K1. R45. ;
¦       :
¦ ┌-  G41 X_ Z_ I_ K_ ;
¦ ¦     :
¦ └-  G40 ;
¦       :
└---  G69.1 ;
```

T

- Relationship between three-dimensional coordinate conversion and tool offset

When using a tool offset command, nest the tool offset command within the three-dimensional coordinate conversion mode.
(Example)

```
┌---  G68.1 X100. Y100. Z100. I0. J0. K1. R45. ;
¦       :
¦ ┌-  T0101;
¦ ¦     :
¦ └-  T0100;
¦       :
└---  G69.1 ;
```

- PMC axis control

In the three-dimensional coordinate conversion mode, PMC axis control cannot be performed for the three axes related to the conversion (alarm).

- Manual operation

When manual feeding is performed during a three-dimensional coordinate conversion, the tangent velocity in the coordinate system after conversion (program coordinate system) equals the lowest feedrate of those on the selected axes.

- Workpiece coordinate system

Avoid changing the workpiece coordinate system in the three-dimensional coordinate conversion mode.

- Manual reference position return

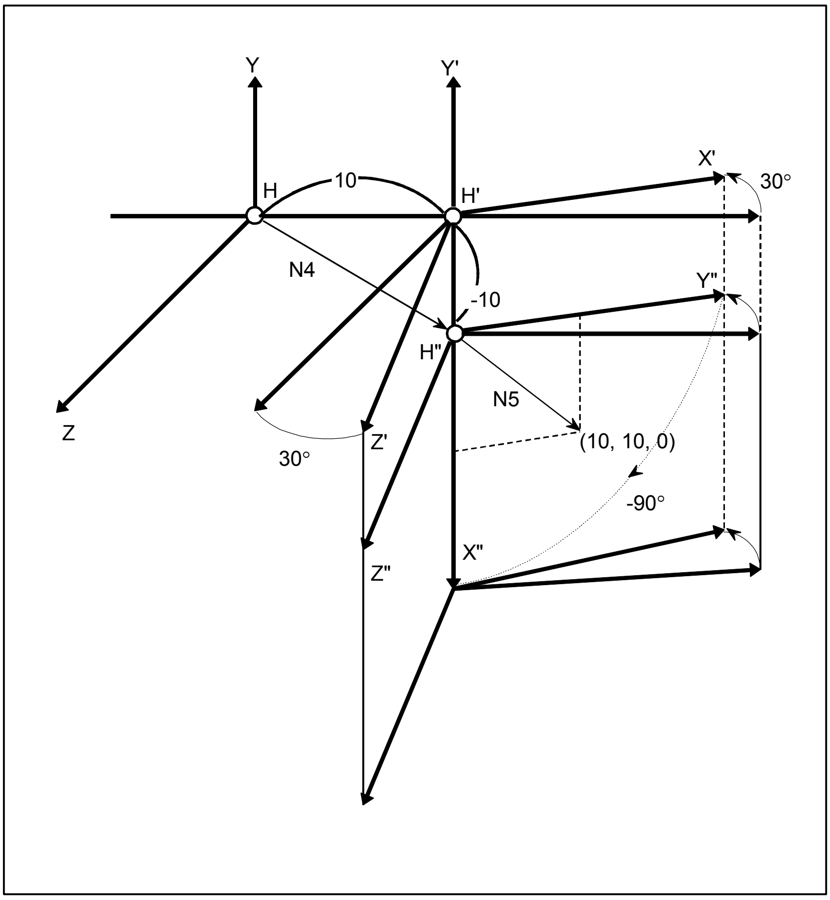
Avoid making a manual reference position return in the three-dimensional coordinate conversion mode.

- Cs contour axis

When specifying the Cs contour axis and rapid traverse at the same time in the three-dimensional coordinate conversion mode, make a reference position return on the Cs contour axis in advance. If a reference position return is made in the first rapid traverse after the Cs contour axis is selected (parameter NRF(No.3700#1) is set to 0), avoid specifying the reference position return command in the three-dimensional coordinate conversion mode.

Example

N1 G90 X0 Y0 Z0 ;	Carries out positioning to zero point H.
N2 G68 X10. Y0 Z0 I0 J1 K0 R30. ;	Forms new coordinate system X'Y'Z'.
N3 G68 X0 Y-10. Z0 I0 J0 K1 R-90. ;	Forms other coordinate system X"Y"Z". The origin agrees with (0, -10, 0) in coordinate system X'Y'Z.
N4 G90 X0 Y0 Z0 ;	Carries out positioning to zero point H" on coordinate system X"Y"Z".
N5 X10. Y10. Z0 ;	Carries out positioning to (10, 10, 0) on coordinate system X"Y"Z".



15 COMPENSATION FUNCTION

This chapter describes the following compensation functions:

15.1 TOOL LENGTH COMPENSATION (G43, G44, G49)

This function can be used by setting the difference between the tool length assumed during programming and the actual tool length of the tool used into the offset memory. It is possible to compensate the difference without changing the program.
Specify the direction of offset with G43 or G44. Select a tool length compensation value from the offset memory by entering the corresponding address and number (H code).

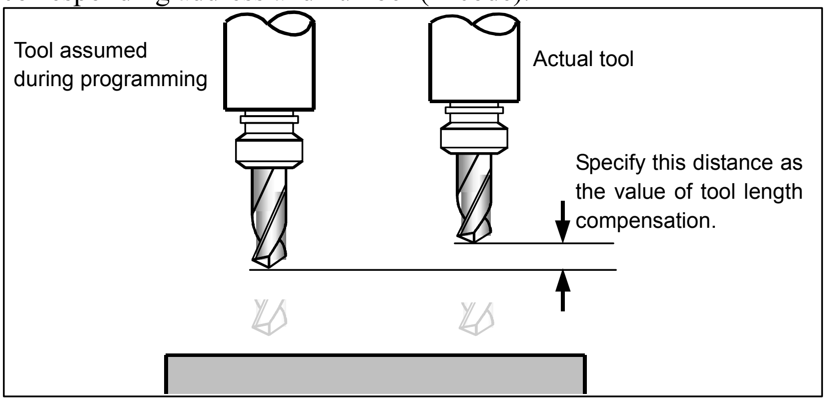


Fig. 15.1 (a) Tool length compensation

15.1.1 Overview

The following three methods of tool length compensation can be used, depending on the axis along which tool length compensation can be made.

- Tool length compensation A
  Compensates for the difference in tool length along the basic Z-axis.
- Tool length compensation B
  Compensates for the difference in tool length in the direction normal to a selected plane.
- Tool length compensation C
  Compensates for the difference in tool length along a specified axis.

Format

Type	Format	Description
Tool length compensation A	**G43 Z_ H_ ;** **G44 Z_ H_ ;**	G43 : Positive offset G44 : Negative offset G17 : XY plane selection G18 : ZX plane selection G19 : YZ plane selection α : Address of a specified axis H : Address for specifying the tool length compensation value X, Y, Z : Offset move command
Tool length compensation B	**G17 G43 Z_ H_ ;** **G17 G44 Z_ H_ ;** **G18 G43 Y_ H_ ;** **G18 G44 Y_ H_ ;** **G19 G43 X_ H_ ;** **G19 G44 X_ H_ ;**	
Tool length compensation C	**G43 α_H_ ;** **G44 α_H_ ;**	
Tool length compensation cancel	**G49 ; or H0 ;**	

Explanation

- Selection of tool length compensation

Select tool length compensation A, B, or C, by setting parameters TLC and TLB (No.5001#0 and #1) .

Parameter No.5001		Type
#1(TLB)	#0(TLC)	
0	0	Tool length compensation A
1	0	Tool length compensation B
0/1	1	Tool length compensation C

- Direction of the offset

When G43 is specified, the tool length compensation value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program. When G44 is specified, the same value is subtracted from the coordinates of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected.
When the specification of an axis is omitted, a movement is made by the tool length compensation value.
G43 and G44 are modal G codes. They are valid until another G code belonging to the same group is used.

- Specification of the tool length compensation value

The tool length compensation value assigned to the number (offset number) specified in the H code is selected from offset memory and added to or subtracted from the moving command in the program.

Example

:	
H1 ;	The offset value of offset number 1 is selected.
:	
G43 Z_ ;	Offset is applied according to the offset value of offset number 1.
:	
H2 ;	Offset is applied according to the offset value of offset number 2.
:	
H0 ;	Offset is applied according to the offset value 0.
:	
H3 ;	Offset is applied according to the offset value of offset number 3.
:	
G49 ;	Offset is canceled.
:	
H4 ;	The offset value of offset number 4 is selected.
:	

A tool length compensation value is to be set in the offset memory corresponding to an offset number.

⚠ **WARNING**

When another offset number is specified, the tool length compensation value just changes to a new value. The new tool length compensation value is not added to the old tool length compensation value.
H1 : Tool length compensation value 20.0
H2 : Tool length compensation value 30.0
G90 G43 Z100.0 H1 ; Z will move to 120.0
G90 G43 Z100.0 H2 ; Z will move to 130.0

NOTE

The tool length compensation value corresponding to offset No. 0, that is, H0 always means 0. It is impossible to set any other tool length compensation value to H0.

- Performing tool length compensation along two or more axes

Tool length compensation B can be executed along two or more axes when the axes are specified in two or more blocks.
By setting bit 3 (TAL) of parameter No. 5001 to 1, cutter compensation C can also can be executed along two or more axes when the axes are specified in two or more blocks. If no axis is specified in the same block, the alarm (PS0027) is issued. If two or more axes are specified in the same block, the alarm (PS0336) is issued.

Example 1

When tool length compensation B is executed along the X-axis and Y-axis

```
G19 G43 H_ ;     Offset in X axis
G18 G43 H_ ;     Offset in Y axis
```

Example 2

When tool length compensation C is executed along the X-axis and Y-axis

```
G43 X_ H_ ;     Offset in X axis
G43 Y_ H_ ;     Offset in Y axis
```

Example 3

When an alarm is issued with tool length compensation C

```
G43 X_ Y_ H_ ;   An alarm (PS0336) occurs
```

- Tool length compensation cancel

To cancel tool length compensation, specify G49 or H0. After G49 or H0 is specified, the system immediately cancels the offset mode.

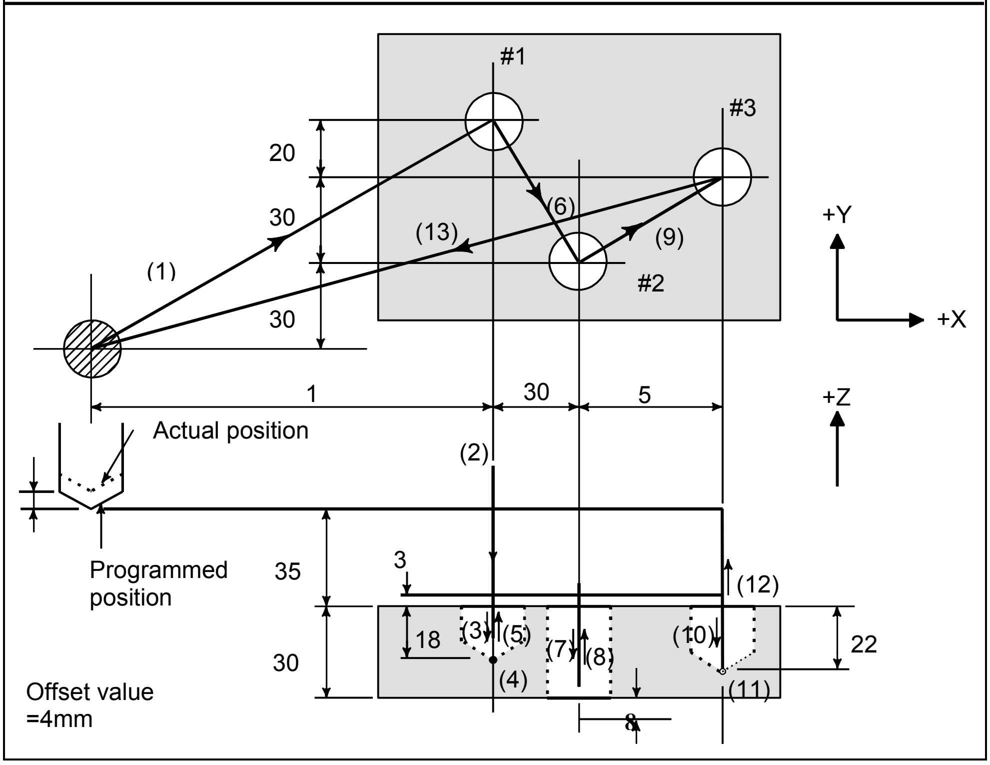
NOTE

1. If offset is executed along two or more axes, offset along all axes is canceled by specifying G49. If H0 is used to specify cancellation, offset along only the axis normal to a selected plane is canceled in the case of tool length compensation B, or offset along only the last axis specified by G43 or G44 is canceled in the case of tool length compensation C.
2. If offset is executed along three or more axes, and offset along all axes is canceled using G49, the alarm (PS0015) (indicating that the number of simultaneously controlled axes is exceeded) may be issued. By using H0 together, for example, cancel offset so that the number of simultaneously controlled axes (the number of axes along which movements are made) does not exceed the allowable range of the system.
3. If tool length compensation and three-dimensional coordinate conversion are canceled by a reset with three-dimensional coordinate conversion performed during tool length compensation, the direction of tool length compensation vector cancellation becomes incorrect. By setting bit 6 (LVK) of parameter No. 5003 to 1 and setting bit 2 (D3R) of parameter No. 5400 to 1, ensure that the tool length compensation vector and three-dimensional coordinate conversion are not canceled by a reset.

Example)
```
G43 H1 ;
G68 X_ Y_ Z_ I_ J_ K_ R_ ;
  :
  :   Ensure that tool length compensation and three-dimensional
  :   coordinate conversion are not canceled by a reset in this range.
G69 ;
G49 ;
```

Example

Tool length compensation (in boring holes #1, #2, and #3)



Program
H1=-4.0 (Tool length compensation value)
N1 G91 G00 X120.0 Y80.0 ;.. (1)
N2 G43 Z-32.0 H1 ; .... (2)
N3 G01 Z-21.0 F1000 ; .... (3)
N4 G04 P2000 ;.. (4)
N5 G00 Z21.0 ;... (5)
N6 X30.0 Y-50.0 ;... (6)
N7 G01 Z-41.0 ; .... (7)
N8 G00 Z41.0 ;... (8)
N9 X50.0 Y30.0 ; .... (9)
N10 G01 Z-25.0 ; .... (10)
N11 G04 P2000 ; .... (11)
N12 G00 Z57.0 H0 ; .... (12)
N13 X-200.0 Y-60.0 ;... (13)
N14 M2 ;

15.1.2 G53, G28, G30, and G30.1 Commands in Tool Length Compensation Mode

This section describes the tool length compensation cancellation and restoration performed when G53, G28, G30, or G31 is specified in tool length compensation mode. Also described is the timing of tool length compensation.

Explanation

- Tool length compensation vector cancellation

When G53, G28, G30, or G30.1 is specified in tool length compensation mode, tool length compensation vectors are canceled as described below. However, the previously specified modal G code remains displayed; modal code display is not switched to G49.

(1) When G53 is specified

Command	Specified axis	Operation
G53 IP_	Tool length compensation axis	Canceled upon movement being performed
	Other than tool length compensation axis	Not canceled
G49 G53 IP_	Tool length compensation axis	Canceled upon movement being performed
	Other than tool length compensation axis	Not canceled

(IP_ : Dimension code)

> ⚠ **CAUTION**
> If tool length compensation is applied along multiple axes, the offset vector along the axis specified by G53 is canceled.

(2) When G28, G30, or G30.1 is specified

Command	Specified axis	Operation
G28 IP_	Tool length compensation axis	Not canceled at an intermediate point. Canceled at the reference position.
	Other than tool length compensation axis	Not canceled at an intermediate point. Canceled at the reference position.
G49 G28 IP_	Tool length compensation axis	Canceled when a movement is made to an intermediate point.
	Other than tool length compensation axis	Canceled when a movement is made to an intermediate point.

(IP_ : Dimension code)

⚠ **CAUTION**
If tool length compensation is applied along multiple axes, the offset vector along the axis on which a reference position return operation has been performed is canceled.

- Tool length compensation vector restoration

Tool length compensation vectors, canceled by specifying G53, G28, G30, or G30.1 in tool length compensation mode, are restored as described below.

Type	Parameter EVO (No.5001#6)	Restoration condition
A/B	0	The H command or G43 (G44) is specified.
	1	Restored by the next buffered block.
C		The H command or G43 (G44)IP_ is specified.

(IP_ : Dimension code)

⚠ **CAUTION**
If a tool length compensation vector is restored only with H_, G43, or G44 when tool length compensation is applied along multiple axes, the tool length compensation vector along only the axis normal to a selected plane is restored in the case of tool length compensation B, or the tool length compensation vector along only the last axis for which tool length compensation is specified is restored in the case of tool length compensation C. The tool length compensation vector along any other axes is not restored.

15.2 SCALING (G50, G51)

Overview

A programmed figure can be magnified or reduced (scaling).
Two types of scaling are available, one in which the same magnification rate is applied to each axis and the other in which different magnification rates are applied to different axes.
The magnification rate can be specified in the program.
Unless specified in the program, the magnification rate specified in the parameter is applied.

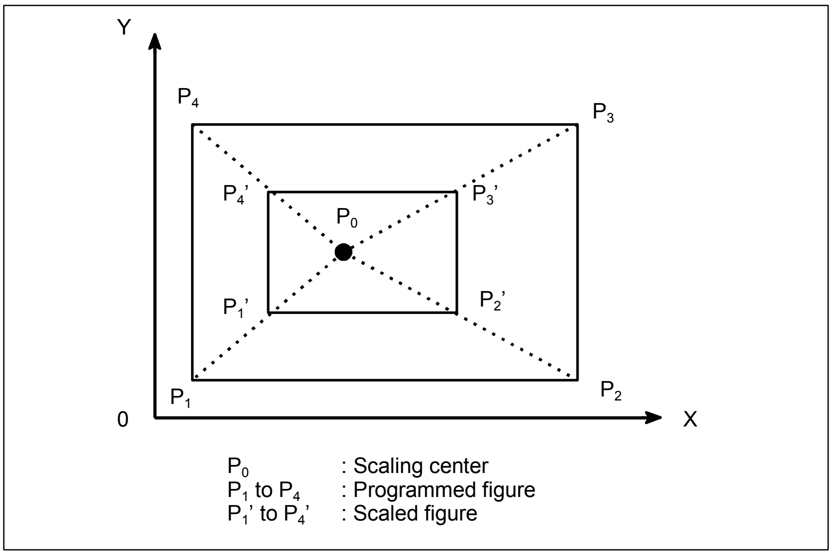


Fig. 15.2 (a) Scaling

Format

Scaling up or down along all axes at the same rate of magnification (When parameter XSC (No. 5400#6) = 0)

Format	Meaning of command
G51 IP_P_ ; Scaling start Scaling is effective. (Scaling mode) **G50 ;** Scaling cancel	**IP_** : Absolute command for center coordinate value of scaling **P_** : Scaling magnification

Scaling up or down along each axes at a different rate of magnification (mirror image) (When parameter XSC (No. 5400#6) = 1)

Format	Meaning of command
G51 IP_I_J_K_; Scaling start Scaling is effective. (Scaling mode) **G50 ;** Scaling cancel	**IP_** : Absolute command for center coordinate value of scaling **I_J_K_** : Scaling magnification for basic 3 axes (X, Y, and Z axes) respectively

T

NOTE
This function is available when the G-code system B or C is set.

⚠ **CAUTION**
1 Specify G51 in a separate block.
2 After the figure is enlarged or reduced, specify G50 to cancel the scaling mode.

NOTE
1 Entering electronic calculator decimal point input mode (parameter DPI (No. 3401#0) = 1) does not cause the units of the magnification rates P, I, J, and K to change.
2 Setting the least input increment equal to 10 times the least command increment (parameter IPR (No. 1004#7) = 1) does not cause the units of the magnification rates P, I, J, and K to change.
3 An attempt to specify 0 as a magnification rate causes alarm (PS0142) to occur in a G51 block.

Explanation

- Axis for which scaling is to be enabled

For the axis for which scaling is to be enabled, set parameter SCL (No. 5401#0) to 1.

- Minimum unit of scaling magnification

Least input increment of scaling magnification is: 0.001 or 0.00001.
It is 0.00001 (one hundred thousandth) if parameter SCR (No. 5400#7) is 0 and 0.001 if it is 1.

- Scaling center

Even in incremental command (G91) mode, the scaling center coordinates IP_ specified in the G51 block are assumed those of an absolute position.
If the scaling center coordinates are omitted, the position assumed when G51 is specified is assumed the scaling center.

> ⚠ **CAUTION**
> With the move command subsequent to the G51 block, execute an absolute (G90 mode) position command.
> If no absolute position command is executed after the G51 block, the position assumed when G51 is specified is assumed the scaling center; once an absolute position command is executed, the scaling center assumes the coordinates specified in the G51 block, after that block.

- Scaling along each axis at the same rate of magnification

Set parameter XSC (No. 5400#6) to 0.
If the scaling magnification P is not specified, the magnification set in parameter (No. 5411) is used.
Decimal point input is not accepted as the magnification P. If decimal point input is made, alarm (PS0007) will occur.
A negative value cannot be specified as the magnification P. If a negative value is specified, alarm (PS0006) will occur.
The allowable magnification range is from 0.00001 to 9999.99999.

- Scaling of each axis, programmable mirror image (negative magnification)

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. The axis subject to the mirror image is the one that contains the scaling center.
Set a parameter XSC (No. 5400#6) to 1 to validate each axis scaling (mirror image).
Using I, J, and K, specify the scaling magnifications for the basic 3 axes (X to Z axes). Use parameter No. 1022 to specify which axes to use as the basic 3 axes. For those of the X to Z axes for which I, J, and K are not specified and for axes other than the basic 3 axes, the magnification set with parameter (No. 5421) is used.
A value other than 0 must be set to parameter (No. 5421).
Decimal point programming can not be used to specify the rate of magnification (I, J, K).
Magnification can be set within the range ±0.00001 ±9999.99999.

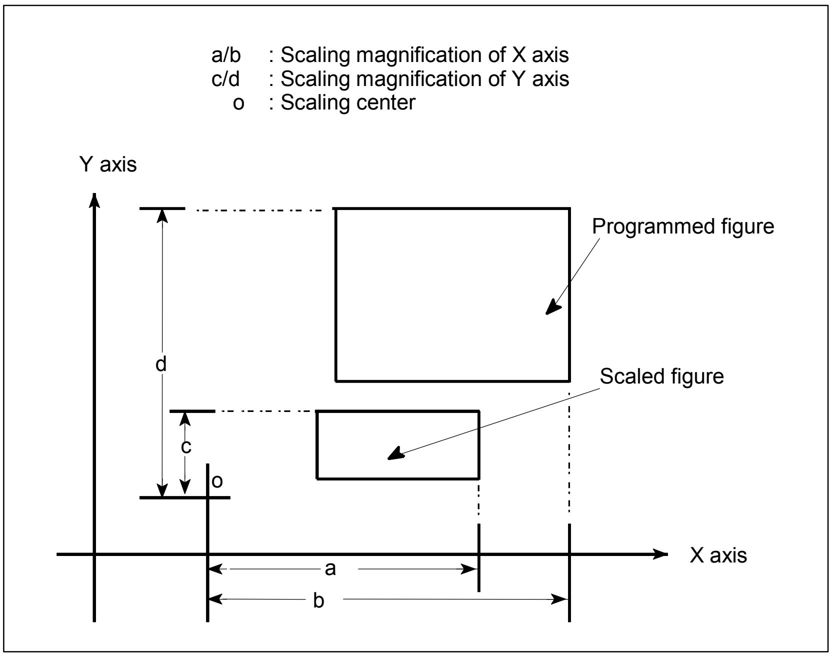


Fig. 15.2 (b) Scaling of each axis

> ⚠ **CAUTION**
> Specifying the following commands at the same time causes them to be executed in the order indicated below:
> <1> Programmable mirror image (G51.1)
> <2> Scaling (G51) (including a mirror image with a negative magnification)
> <3> Mirror image due to the external switch of the CNC or the settings of the CNC
> In this case, the programmable mirror image is effective to the scaling center and magnification as well.
> To specify G51.1 and G51 at the same time, specify them in this order; to cancel them, specify them in the reverse order.

- Scaling of circular interpolation

Even if different magnifications are applied to each axis in circular interpolation, the tool will not trace an ellipse.

G90 G00 X0.0 Y100.0 Z0.0;
G51 X0.0 Y0.0 Z0.0 I2000 J1000;
(A magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.)
G02 X100.0 Y0.0 I0 J–100.0 F500;

Above commands are equivalent to the following command:

G90 G00 X0.0 Y100.0 Z0.0;
G02 X200.0 Y0.0 I0 J–100.0 F500;
(Because the end point is not on an arc, spiral interpolation is assumed.)

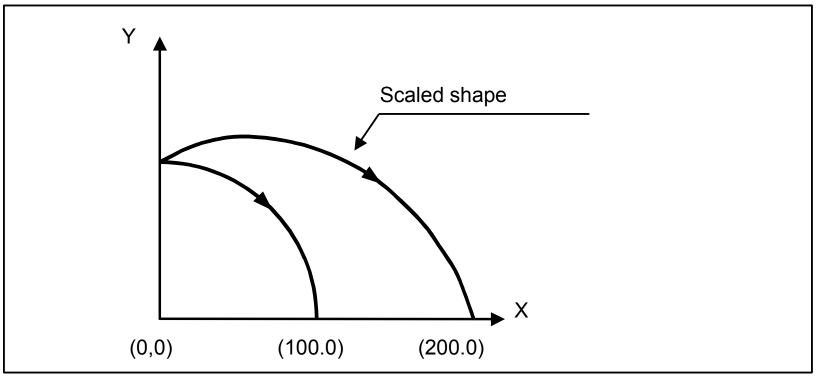


Fig. 15.2 (c) Scaling for circular interpolation1

Even for an R-specified arc, scaling is applied to each of I, J, and K after the radius value (R) is converted into a vector in the center direction of each axis.

If, therefore, the above G02 block contains the following R-specified arc, the operation will be same as that in which I and J are specified.

G02 X100.0 Y0.0 R100.0 F500 ;

- Scaling and coordinate system rotation

If both scaling and coordinate system rotation are specified at the same time, scaling is performed first, followed by coordinate system rotation. In this case, scaling is effective to the rotation center as well.
To specify both of them, specify scaling first and then coordinate system rotation. To cancel them, specify them in the reverse order.

Example
Main program

```
O1
G90 G00 X20.0 Y10.0 ;
M98 P1000 ;
G51 X20.0 Y10.0 I3000 J2000 ; (x 3 in the X direction and x 2
                                in the Y direction)
M98 P1000 ;
G17 G68 X35.0 Y20.0 R30. ;
M98 P1000 ;
G69 ;
G50 ;
M30 ;
```

Subprogram

```
O1000 ;
G01 X20.0 Y10.0 F500 ;
G01 X50.0 ;
G01 Y30.0 ;
G01 X20.0 ;
G01 Y10.0 ;
M99 ;
```

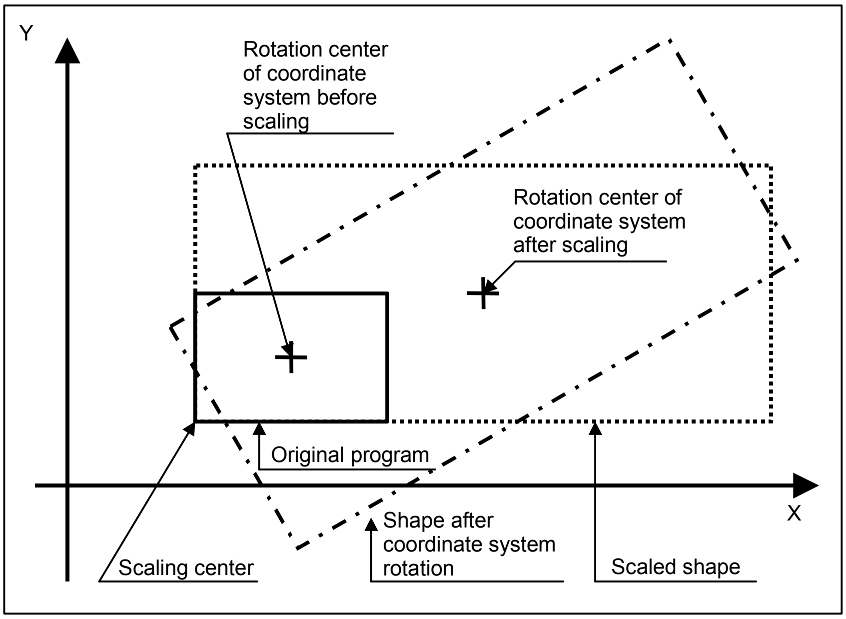


Fig. 15.2 (d) Scaling and coordinate system rotation

- Scaling and optional chamfering/corner R

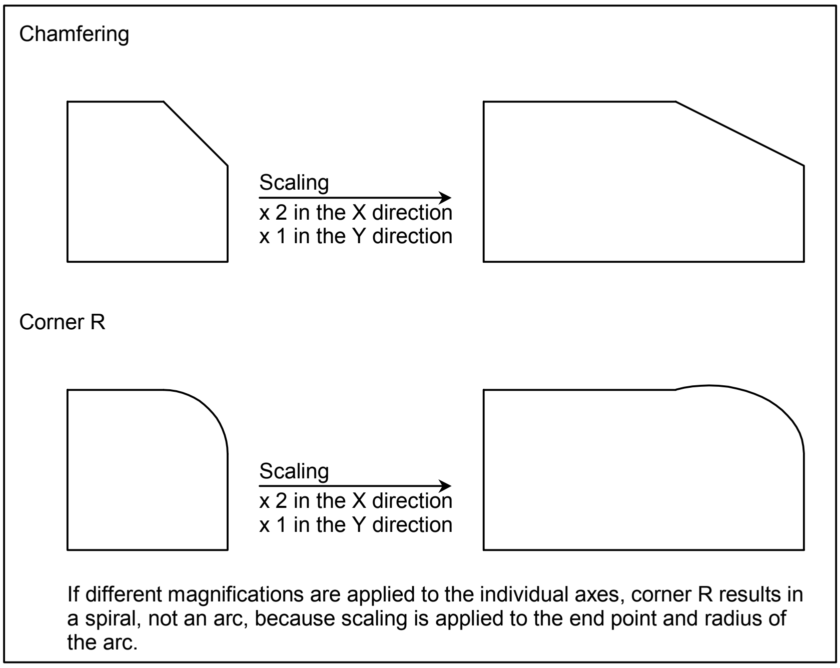


Fig. 15.2 (e) Scaling and optional chamfering/corner R

Limitation

- Tool compensation

This scaling is not applicable to cutter or tool nose radius compensation values, tool length compensation values, and tool offset values (Fig. 15.2 (f)).

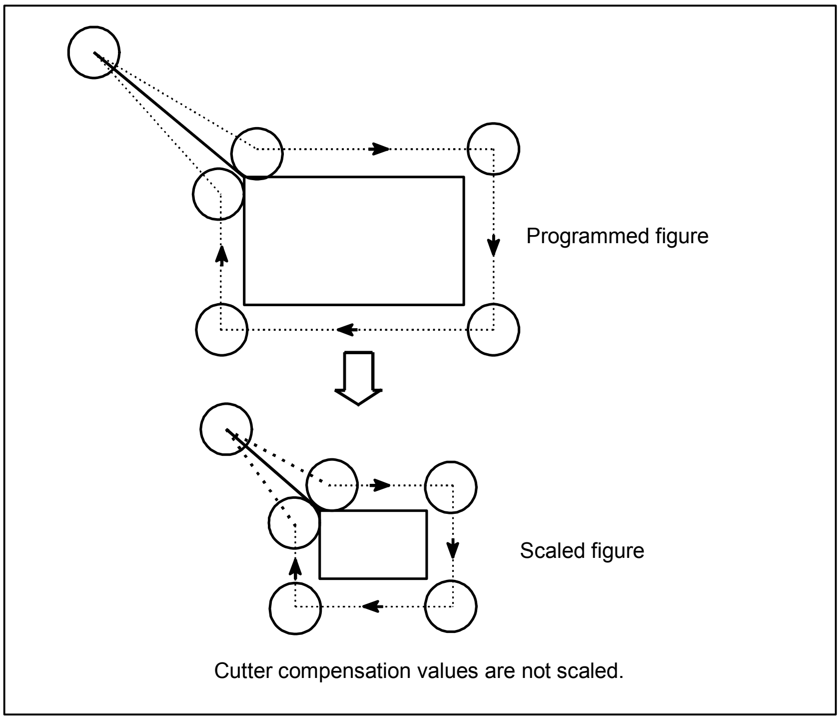


Fig. 15.2 (f) Scaling during cutter compensation

- Invalid scaling

M

Scaling is not applied to the travel distance during canned cycle shown below.

- Cut-in value Q and retraction value d of peck drilling cycle (G83, G73).
- Fine boring cycle (G76)
- Shift value Q of X and Y axes in back boring cycle (G87).

In manual operation, the travel distance cannot be increased or decreased using the scaling function.

T

This function is available to G code systems B and C only; it is not available to G code system A.
During scaling, the following functions cannot be used. If any of them is specified, alarm PS0300 will occur.

- Finishing cycle (G70, G72)
- Outer surface rough cutting cycle (G71, G73)
- End rough cutting cycle (G72, G74)
- Closed loop cutting cycle (G73, G75)
- End cutting off cycle (G74, G76)
- Outer/inner surface cutting off cycle (G75, G77)
- Multiple threading cycle (G76, G78)
- Traverse grinding cycle (for grinding machines) (G71, G72)
- Traverse direct constant-size grinding cycle (for grinding machines) (G72, G73)
- Oscillation grinding cycle (for grinding machines) (G73, G74)
- Oscillation direct constant-size grinding cycle (G74, G75)
- Face drilling cycle (G83, G83)
- Face tapping cycle (G84, G84)
- Face boring cycle (G85, G85)
- Side drilling cycle (G87, G87)
- Side tapping cycle (G88, G88)
- Side boring cycle (G89, G89)
- Outer/inner surface turning cycle (G77, G20)
- Threading cycle (G78, G21)
- End turning cycle (G79, G24)

(The G code in system B is given first, followed by that of system C.)

Scaling is not applied to the travel distance during manual operation.

⚠ CAUTION

1 If a parameter setting value is employed as a scaling magnification without specifying P, the setting value at G51 command time is employed as the scaling magnification, and a change of this value, if any, is not effective.
2 Before specifying the G code for reference position return (G27, G28, G29, G30) or coordinate system setting (G52 to G59, G92), cancel the scaling mode.
3 If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may become zero. In this case, the block is regarded as a no movement block, and therefore, it may affect the tool movement by cutter compensation. See the description of cutter compensation.
4 Refrain from scaling on a rotation axis for which the rollover function is enabled. Otherwise, the tool may rotate in a short-cut manner, possibly resulting in unexpected movement.

NOTE

1 The position display represents the coordinate value after scaling.
2 When a mirror image was applied to one axis of the specified plane, the following results:
   (1) Circular command
   .... Direction of rotation is reversed.
   (2) Cutter or tool nose radius compensation
   .... Offset direction is reversed.
   (3) Coordinate system rotation
   .... Rotation angle is reversed.

Example

Sample program of a scaling in each axis

```
O1;
G51 X20.0 Y10.0 I750 J250; (× 0.75 in the X direction, × 0.25 in
                            the Y direction)
G00 G90 X60.0 Y50.0;
G01 X120.0 F100;
G01 Y90;
G01 X60;
G01 Y50;
G50;
M30;
```

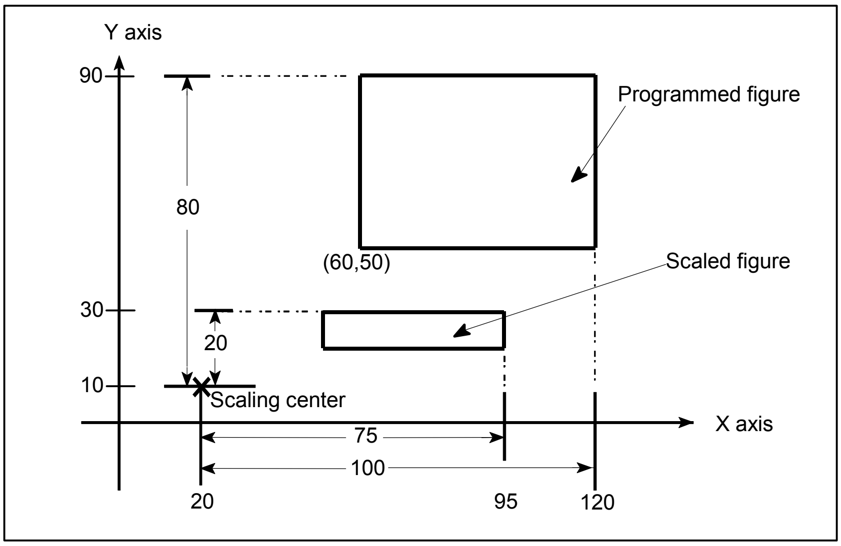
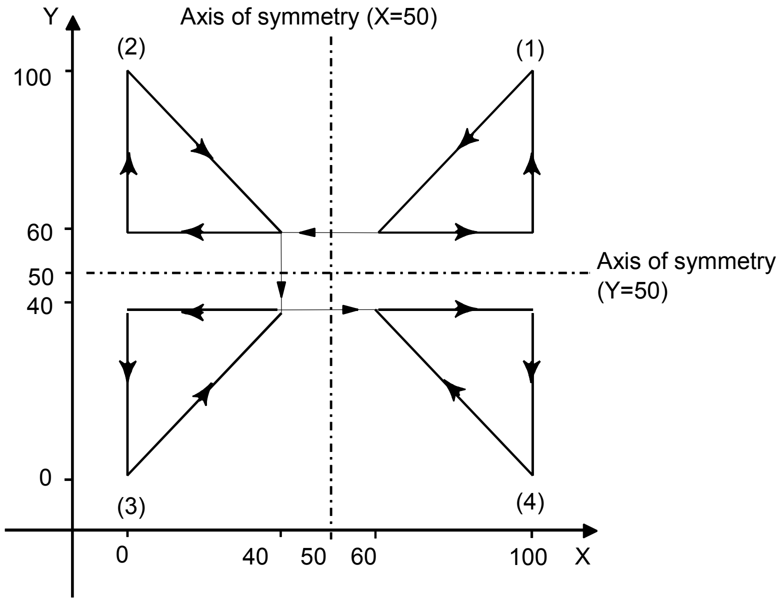


Fig. 15.2 (g) Program example of scaling in each axis

15.3 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1)

A mirror image of a programmed command can be produced with respect to a programmed axis of symmetry (Fig. 15.3 (a)).



(1) Original image of a programmed command
(2) Image symmetrical about a line parallel to the Y-axis and crossing the X-axis at 50
(3) Image symmetrical about point (50, 50)
(4) Image symmetrical about a line parallel to the X-axis and crossing the Y-axis at 50

Fig. 15.3 (a) Programmable mirror image

Format

G51.1 IP_ ;	Setting a programmable image
:	A mirror image of a command specified in these blocks is produced with respect to the axis of symmetry specified by G51.1 IP_;.
:	
:	
:	
G50.1 IP_ ;	Canceling a programmable mirror image
IP_ :	Point (position) and axis of symmetry for producing a mirror image when specified with G51.1. Axis of symmetry for producing a mirror image when specified with G50.1. Point of symmetry is not specified.

Explanation

- Mirror image by setting

If the programmable mirror image function is specified when the command for producing a mirror image is also selected by a CNC external switch or CNC setting (see III-4.5), the programmable mirror image function is executed first.

- Mirror image on a single axis in a specified plane

Applying a mirror image to one of the axes on a specified plane changes the following commands as follows :

Command	Explanation
Circular command	G02 and G03 are interchanged.
Cutter or tool nose radius compensation	G41 and G42 are interchanged.
Coordinate system rotation	CW and CCW (directions of rotation) are interchanged.

Limitation

- Scaling and coordinate system rotation

Processing proceeds from program mirror image to scaling and coordinate system rotation in the stated order. The commands should be specified in this order, and, for cancellation, in the reverse order. Do not specify G50.1 or G51.1 during scaling or coordinate system rotation mode.

- Commands related to reference position return and coordinate system

In programmable mirror image mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling the programmable mirror image mode.

15.4 TOOL AXIS DIRECTION TOOL LENGTH COMPENSATION

Overview

When a five-axis machine that has two axes for rotating the tool is used, tool length compensation can be performed in a specified tool axis direction on a rotation axis. When a rotation axis is specified in tool axis direction tool length compensation mode, tool length compensation is applied in a specified tool axis direction on the rotation axis by the compensation value specified in the H code. That is, movement is made along the three linear axes (Xp, Yp, Zp).Unless otherwise noted in the explanation of this function, the two rotation axes are assumed to be the B-axis and C-axis.

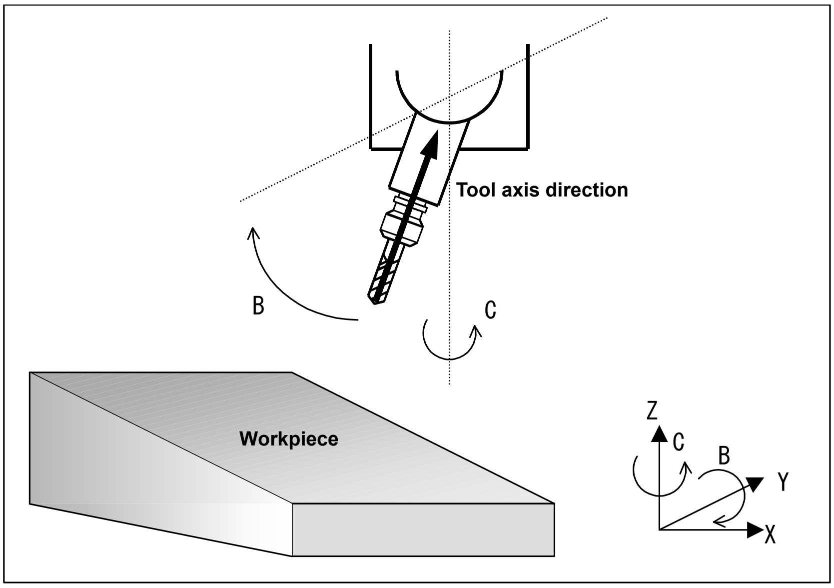


Fig. 15.4(a) Tool Axis Direction Tool Length Compensation

Format

- Tool axis direction tool length compensation command

G43.1 Hn ;
n: Compensation number

- Tool axis direction tool length compensation cancel command

G49 ;

Explanation

- Command for tool axis direction tool length compensation

The tool compensation vector changes as the offset value changes or movement is made on a rotation axis. When the tool compensation vector changes, movement is made according to the change value along the X-axis, Y-axis, and Z-axis.
When the command specifies movement on a rotation axis only, the position of the tool tip is the same both before and after execution of

the command. (During rotation axis movement, however, the tool tip moves.)

- Examples of machine configuration and rotation axis calculation formats

Let Vx, Vy, Vz, Lc, a, b, and c be as follows :

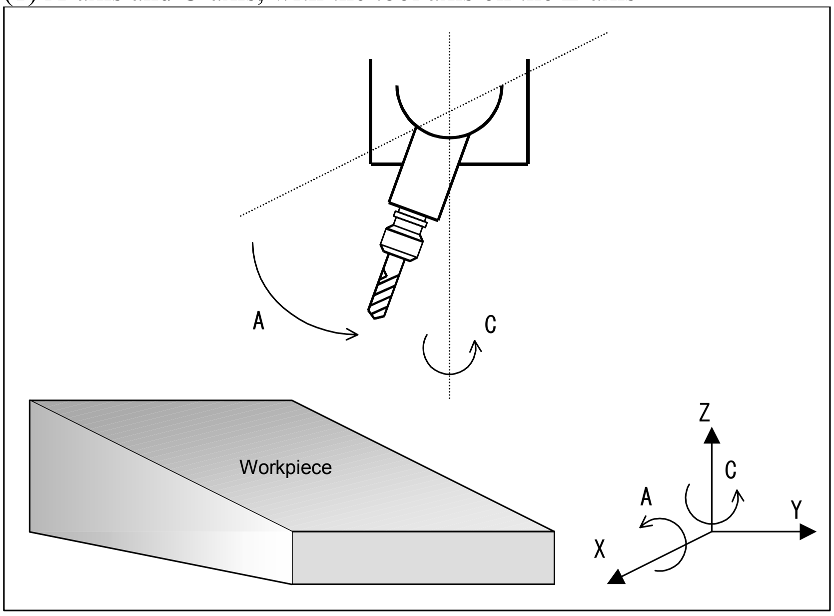
Vx,Vy,Vz : Tool compensation vectors along the X-axis, Y-axis, and Z-axis

Lc : Offset value

a,b,c : Absolute coordinates on the A-axis, B-axis and Caxis

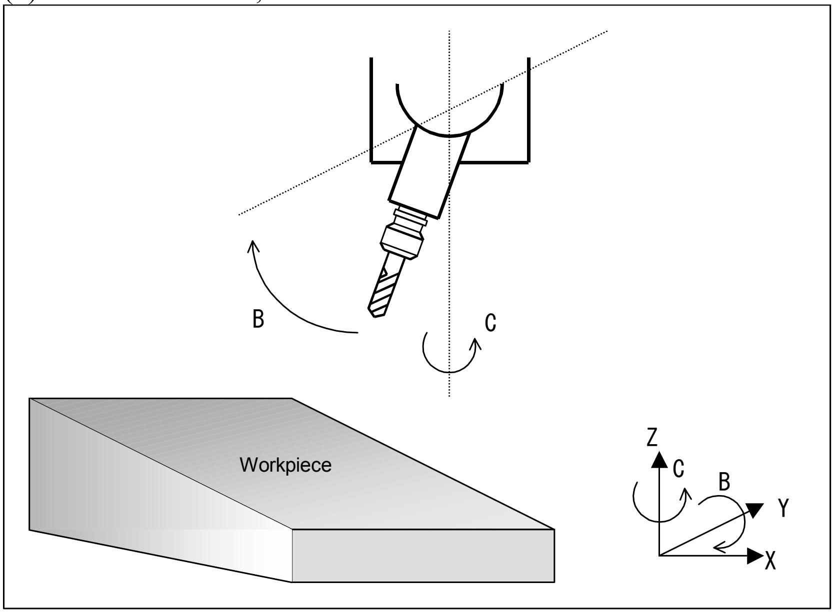
Then, the tool compensation vector on each axis in each machine configuration is indicated below.

(1) A-axis and C-axis, with the tool axis on the Z-axis



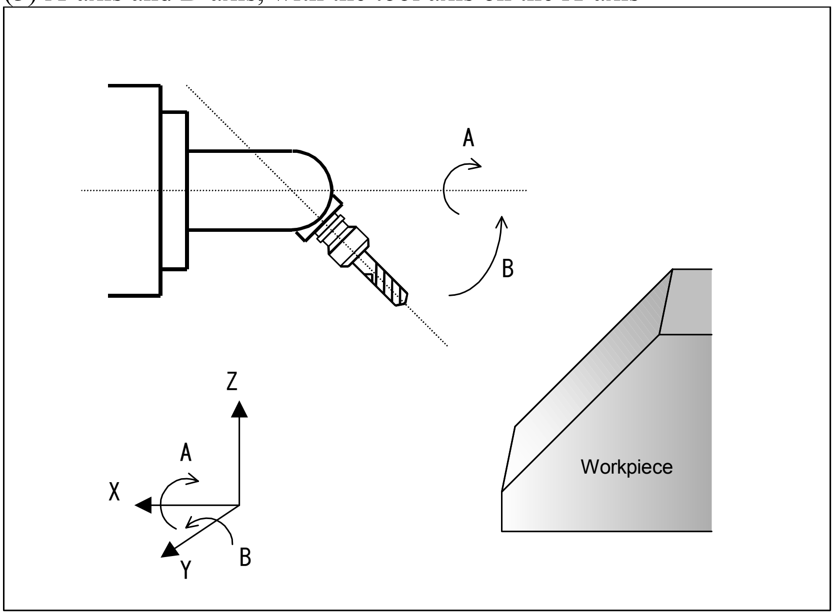
Vx = Lc * sin(a) * sin(c)
Vy = -Lc * sin(a) * cos(c)
Vz = Lc * cos(a)

(2) B-axis and C-axis, with the tool axis on the Z-axis



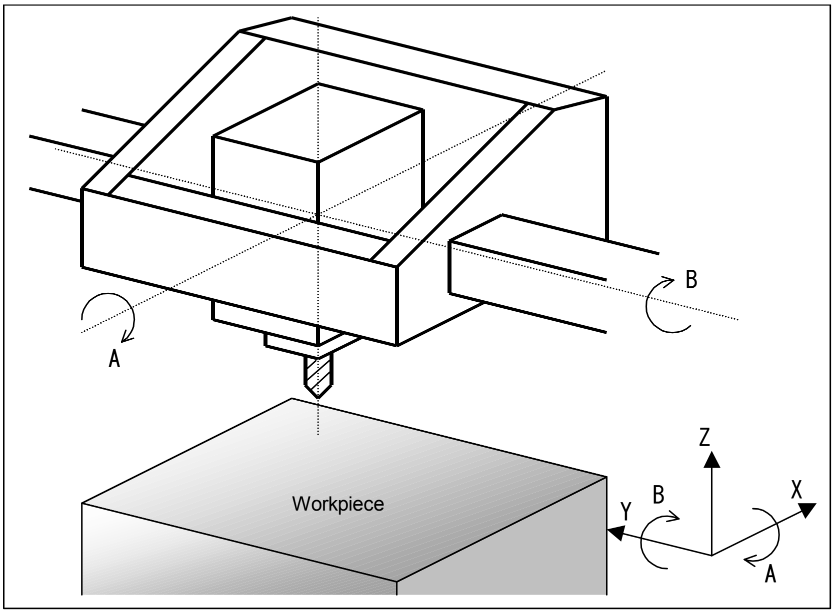
Vx = Lc * sin(b) * cos(c)
Vy = Lc * sin(b) * sin(c)
Vz = Lc * cos(b)

(3) A-axis and B-axis, with the tool axis on the X-axis



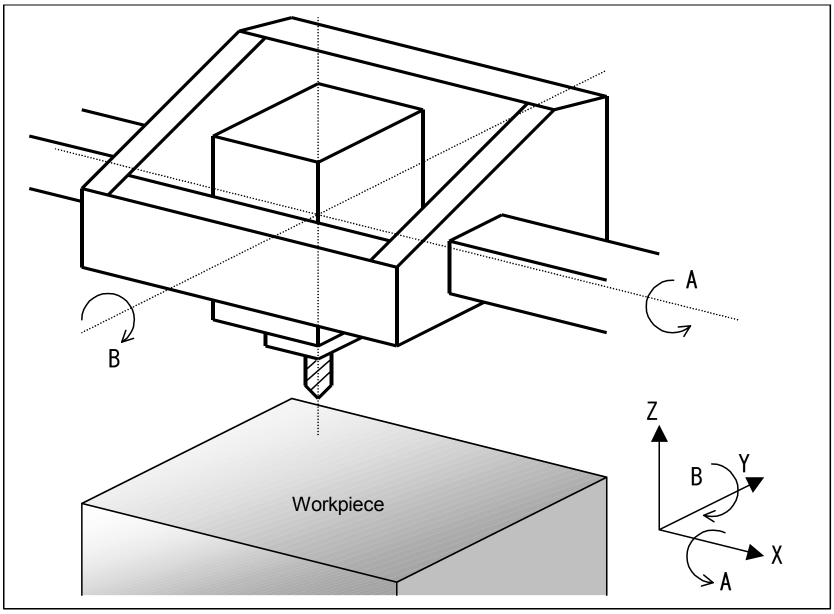
Vx = Lc * cos(b)
Vy = Lc * sin(b) * sin(a)
Vz = -Lc * sin(b) * cos(a)

(4) A-axis and B-axis, with the tool axis on the Z-axis, and the B-axis used as the master



Vx = Lc * cos(a) * sin(b)
Vy = -Lc * sin(a)
Vz = Lc * cos(a) * cos(b)

(5) A-axis and B-axis, with the tool axis on the Z-axis, and the A-axis used as the master



Vx = Lc * sin(b)
Vy = -Lc * sin(a) * cos(b)
Vz = Lc * cos(a) * cos(b)

- Tool holder offset

The machine-specific length from the rotation center of the tool rotation axes (A- and B-axes, A- and C-axes, and B- and C-axes) to the tool mounting position is referred to as the tool holder offset. Unlike a tool length offset value, a tool holder offset value is set in parameter No.19666. When tool axis direction tool length compensation is applied, the sum of the tool holder offset and tool length offset is handled as a tool length for compensation calculation.

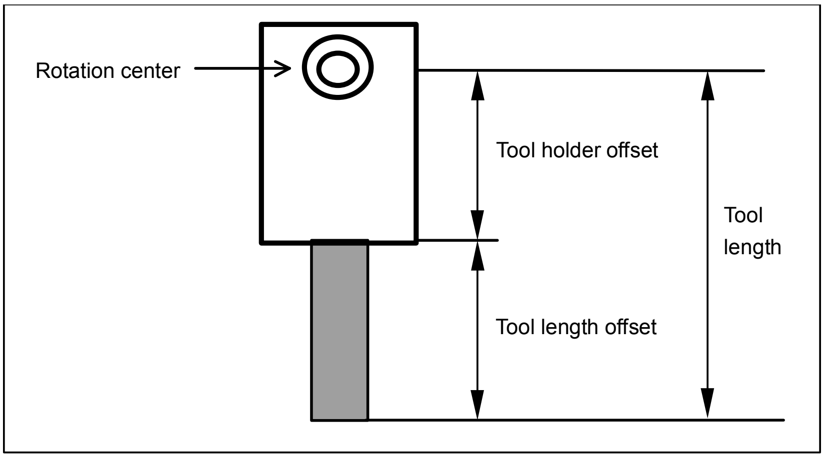


Fig. 15.4(b) Tool Holder Offset

- Parameter-based rotation angle specification

A tool compensation vector is found from the coordinates on the rotation axes for controlling the tool axis direction. However, the configuration of some machines is such that the tool axis is inclined using a fixed attachment. In such a case, the rotation angles of the rotation axes can be set using parameters.
Set bit 1 (RAP) of parameter No. 19650 to 1, and set the coordinates in parameter No. 19658.

- Rotation axis origin compensation

This function compensates for a slight shift of the rotation axis origin caused, for example, by thermal displacement. Specify a compensation value in parameter No. 19660.
When the tool axis is on the Z-axis, and the rotation axes are the B-axis and C-axis, a compensation vector is calculated as follows :

Xp = Lc * sin(B-Bz) * cos(C-Cz)
Yp = Lc * sin(B-Bz) * sin(C-Cz)
Zp = Lc * cos(B-Bz)

Xp,Yp,Zp : Compensation pulse on each axis after origin shift compensation
Lc : Offset value
B,C : Machine position on B-axis and C-axis
Bz,Cz : Origin compensation value on B-axis and C-axis

- Rotation axis offset

Set offsets relative to the rotation angles of the rotation axes in parameter No. 19659. The compensation vector calculation formula is the same as that used for rotation axis origin compensation, except that Bp and Cp are changed to rotation axis offsets.

When rotation axis origin compensation and rotation offsetting are set at the same time, both compensations are performed.
When the tool axis is on the Z-axis, and the rotation axes are the B-axis
and C-axis, compensation vector calculation is performed as follows :

Xp = Lc * sin(B-(Bz+Bo)) * cos(C-(Cz+Co))
Yp = Lc * sin(B-(Bz+Bo)) * sin(C-(Cz+Co))
Zp = Lc * cos(B-(Bz+Bo))
Bz,Cz : B-axis and C-axis origin compensation values
Bo,Co : B-axis and C-axis rotation axis offset values

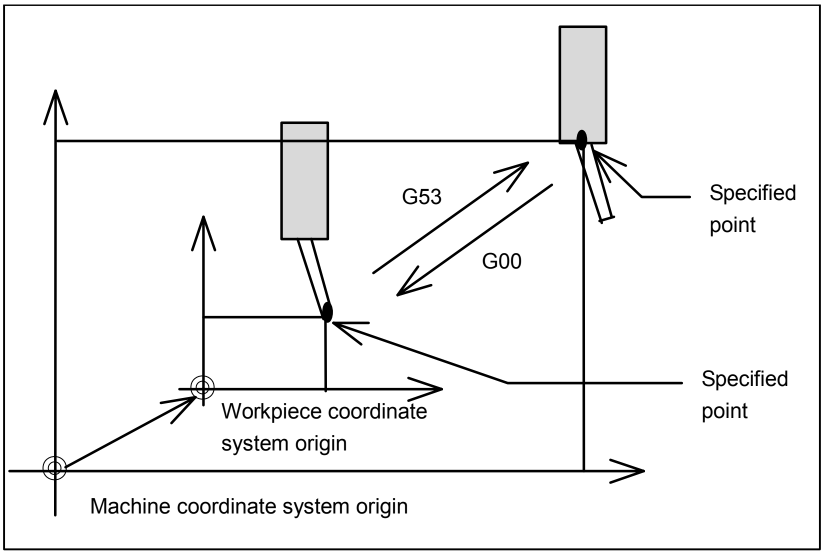
Limitation

- Automatic reference position return command (G28, G29, G30)

Never specify an automatic reference position return command (G28, G29, or G30) in tool axis direction tool length compensation mode.
If an automatic reference position return command is specified in tool axis direction tool length compensation mode, the compensation vector is cancelled at the time of reference position return. So, correct tool axis direction tool length compensation is not performed in subsequent movement along linear axes.

- Machine coordinate system positioning (G53)

When machine coordinate system positioning (G53) is performed, the compensation vector is temporarily cancelled in the block, but is applied when movement is next performed.



15.4.1 Control Point Compensation of Tool Length Compensation Along Tool Axis

Normally, the control point of tool length compensation along the tool axis is the point of intersection of the centers of two rotation axes. The machine coordinates also indicate this control point.
This section explains the compensation performed when the centers of the two rotation axes do not intersect and also explains how to place the control point at a convenient position on the machine.

Explanation

- Compensation of the rotation centers of two rotation axes

Compensation when the rotation centers of two rotation axes do not match is performed.
The length from the tool mounting position to the first rotation axis center is set as the tool holder offset value in parameter No. 19666.
The vector from the first rotation axis center to the second rotation axis center is set as the rotation center compensation vector in parameter No. 19661. Since parameter No. 19661 is an axis type parameter, the compensation amount for three axes (X, Y, and Z) can be set in this parameter.

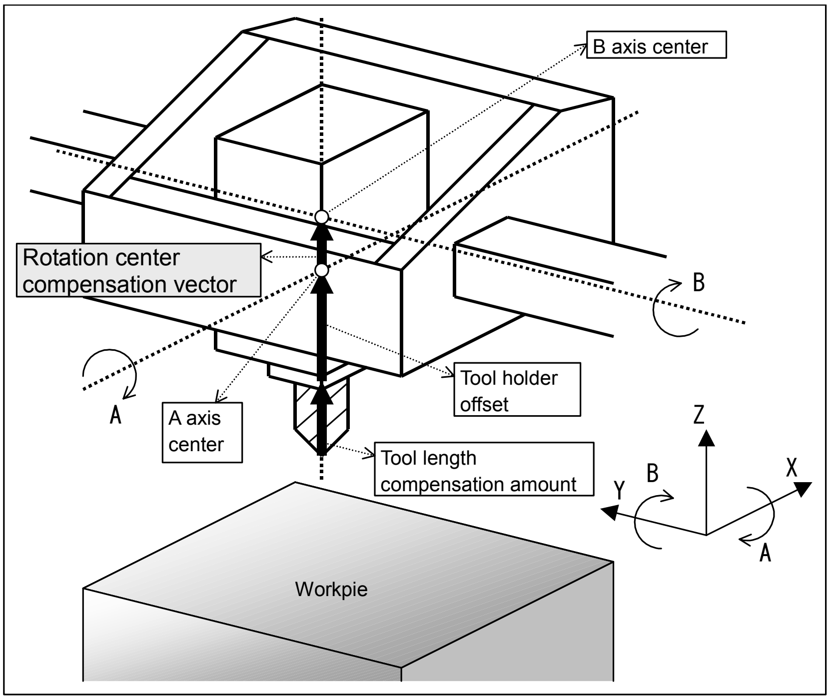


Fig. 15.4(c) Compensation of Rotation Centers of Two Rotation Axes

According to the machine type, set the values listed in the following table:

Table 15.4(a) Setting the Tool Holder Offset and Rotation Center Compensation Vector

Machine type	Tool holder offset Parameter No. 19666	Rotation center compensation vector Parameter No. 19661
(1) A- and C- axes. Tool axis is Z-axis.	Length from tool mounting position to A-axis center	Vector from A-axis center to C-axis center
(2) B- and C- axes. Tool axis is Z-axis.	Length from tool mounting position to B-axis center	Vector from B-axis center to C-axis center
(3) A- and B- axes. Tool axis is X-axis.	Length from tool mounting position to B-axis center	Vector from B-axis center to A-axis center
(4) A- and B- axes. Tool axis is Z-axis. B-axis is master.	Length from tool mounting position to A-axis center	Vector from A-axis center to B-axis center
(5) A- and B-axes. Tool axis is Z-axis. A-axis is master	Length from tool mounting position to B-axis center	Vector from B-axis center to A-axis center

NOTE

When using the spindle center compensation described blow, set the length from the tool mounting position to the spindle center as the tool holder offset.

- Spindle center compensation

Compensation of the spindle center is performed.
The amount of spindle center compensation is set in parameter No. 19662. Since parameter No. 19662 is an axis type parameter, the compensation amount for three axes (X, Y, and Z) can be set in this parameter.

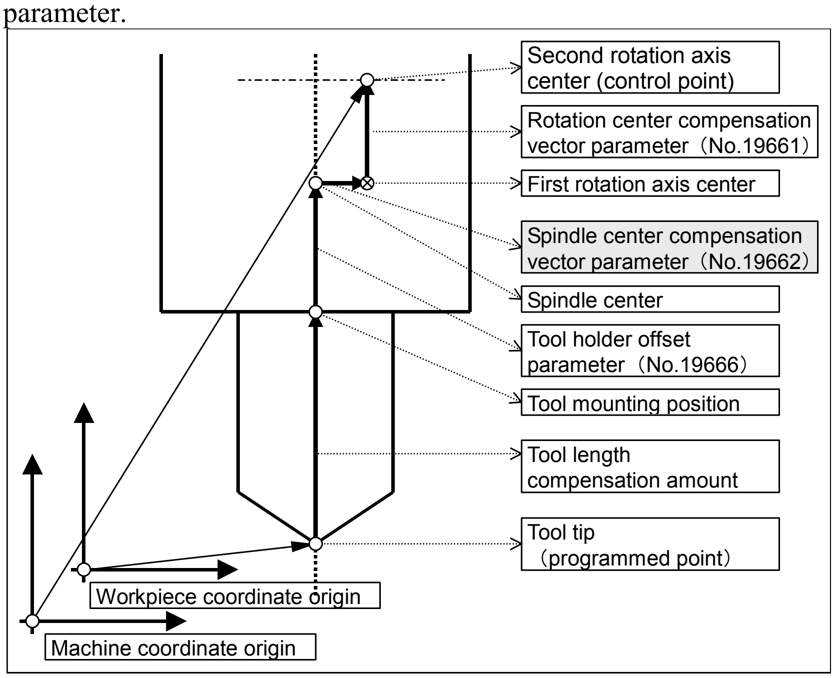


Fig. 15.4(d) Spindle Center Compensation

- Shifting the control point

Conventionally, the center of a rotation axis was used as the control point. The control point can now be shifted as shown in the figure below.

Then, when the rotation axis is at the 0-degree position also in tool length compensation along the tool axis (G43.1), the control point can be set to the same position as that for ordinary tool length compensation (G43).

The control point here is indicated with machine coordinates.

When linear interpolation is specified, for example, this control point moves linearly.

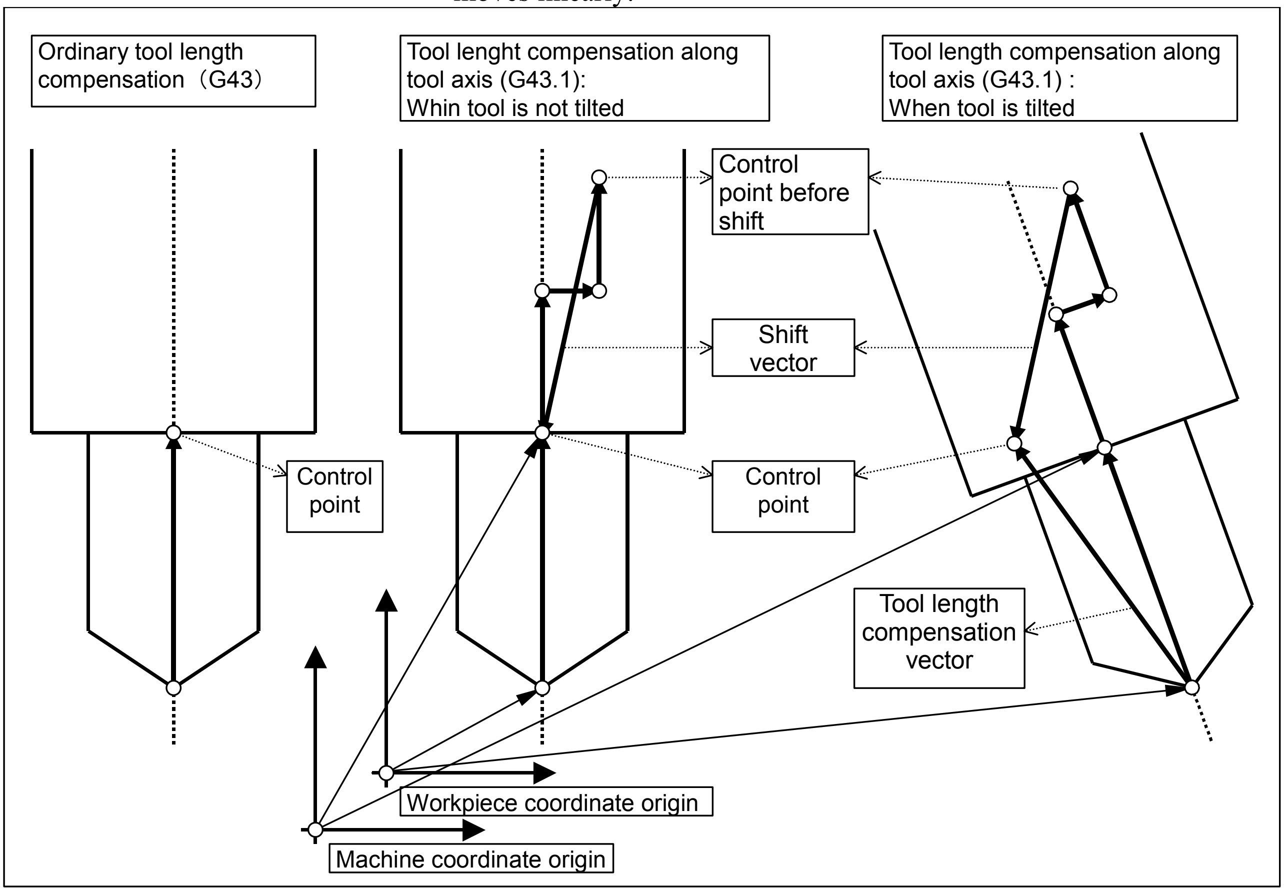


Fig. 15.4(e) Shift of the Control Point

The method of shifting the control point can be selected using the following parameters:

Table 15.4(b) Methods of Shifting the Control Point

SVC (bit 5 of parameter No. 19665)	SPR (bit 4 of parameter No. 19665)	Shift of controlled point
0	-	Shift is not performed an not done conventionally
1	0	The controlled point is shifted according to the result of the following automatic calculation: -（intersection offset vector between the tool axis and the first rotation axis of the tool + intersection offset vector between the second and first rotation axes of the tool offset + tool holder offset (parameter No. 19666)）
1	1	The controlled point is shifted. As the shift vector, the vector set in parameter No. 19667 is used.

- Equation for each machine type

Vx, Vy, Vz :Tool length compensation vector
A, B, C :Absolute coordinates of A-, B-, and C-axes
To :Tool offset value
Ho :Tool holder offset value
Jx, Jy, Jz :Rotation center compensation vector
Cx, Cy, Cz :Spindle center compensation vector
Sx, Sy, Sz: Shift vector

Suppose the above. Then, the tool length compensation vector for each axis is calculated depending on the machine type, as follows:

(1) A- and C-axes. The tool axis is the Zaxis.

$$\begin{bmatrix} Vx \\ Vy \\ Vz \end{bmatrix} = \begin{bmatrix} \cos C & -\sin C & 0 \\ \sin C & \cos C & 0 \\ 0 & 0 & 1 \end{bmatrix} \left(\begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos A & -\sin A \\ 0 & \sin A & \cos A \end{bmatrix} \begin{bmatrix} Cx \\ Cy \\ To + Ho + Cz \end{bmatrix} + \begin{bmatrix} Jx \\ Jy \\ Jz \end{bmatrix} \right) + \begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix}$$

(2) B- and C-axes. The tool axis is the Z-axis.

$$\begin{bmatrix} Vx \\ Vy \\ Vz \end{bmatrix} = \begin{bmatrix} \cos C & -\sin C & 0 \\ \sin C & \cos C & 0 \\ 0 & 0 & 1 \end{bmatrix} \left(\begin{bmatrix} \cos B & 0 & \sin B \\ 0 & 1 & 0 \\ -\sin B & 0 & \cos B \end{bmatrix} \begin{bmatrix} Cx \\ Cy \\ To + Ho + Cz \end{bmatrix} + \begin{bmatrix} Jx \\ Jy \\ Jz \end{bmatrix} \right) + \begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix}$$

(3) A- and B-axes The tool axis is the X-axis.

$$\begin{bmatrix} Vx \\ Vy \\ Vz \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos A & -\sin A \\ 0 & \sin A & \cos A \end{bmatrix} \left(\begin{bmatrix} \cos B & 0 & \sin B \\ 0 & 1 & 0 \\ -\sin B & 0 & \cos B \end{bmatrix} \begin{bmatrix} To + Ho + Cx \\ Cy \\ Cz \end{bmatrix} + \begin{bmatrix} Jx \\ Jy \\ Jz \end{bmatrix} \right) + \begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix}$$

(4) A- and B-axes. The toolaxis is the Z-axis, and the B-axis is the master.

$$\begin{bmatrix} Vx \\ Vy \\ Vz \end{bmatrix} = \begin{bmatrix} \cos B & 0 & \sin B \\ 0 & 1 & 0 \\ -\sin B & 0 & \cos B \end{bmatrix} \left(\begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos A & -\sin A \\ 0 & \sin A & \cos A \end{bmatrix} \begin{bmatrix} Cx \\ Cy \\ To + Ho + Cz \end{bmatrix} + \begin{bmatrix} Jx \\ Jy \\ Jz \end{bmatrix} \right) + \begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix}$$

(5) A- and B-axes. The tool axis is the Z-axis and the A-axis is the master.

$$\begin{bmatrix} Vx \\ Vy \\ Vz \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos A & -\sin A \\ 0 & \sin A & \cos A \end{bmatrix} \left(\begin{bmatrix} \cos B & 0 & \sin B \\ 0 & 1 & 0 \\ -\sin B & 0 & \cos B \end{bmatrix} \begin{bmatrix} Cx \\ Cy \\ To + Ho + Cz \end{bmatrix} + \begin{bmatrix} Jx \\ Jy \\ Jz \end{bmatrix} \right) + \begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix}$$

The shift vector (Sx, Sy, Sz) is calculated as follows:

(A) When bit 5 (SVC) of parameter No. 19665 = 0, the vector is set to 0.

(B) When bit 5 (SVC) of parameter No. 19665 = 1, and bit 4 (SBP) of parameter No. 19665 = 0:

When the machine type is other than (3)

$$\begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix} = -\begin{bmatrix} Cx + Jx \\ Cy + Jy \\ Cz + Jz + Ho \end{bmatrix}$$

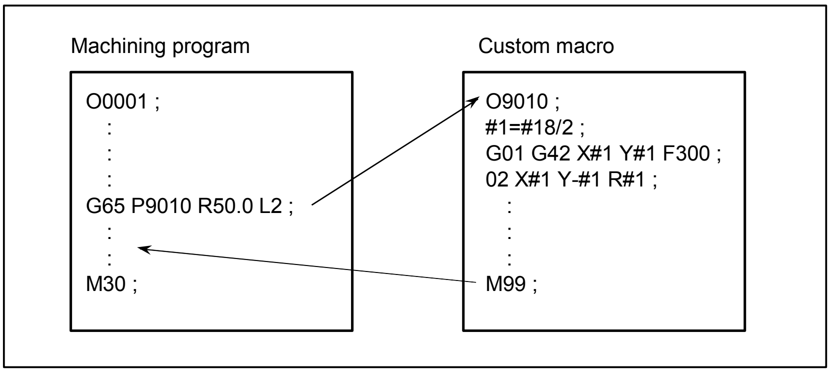
When the machine type is (3)

$$\begin{bmatrix} Sx \\ Sy \\ Sz \end{bmatrix} = -\begin{bmatrix} Cx + Jx + Ho \\ Cy + Jy \\ Cz + Jz \end{bmatrix}$$

(C) When bit 5 (SVC) of parameter No. 19665 = 1, and bit 4 (SBP) of parameter No. 19665 = 1, the vector specified in parameter No. 19667 is set.

16 CUSTOM MACRO

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user-defined canned cycles.
A machining program can call a custom macro with a simple command, just like a subprogram.



16.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 and X100.0.
With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI panel.

```
#1=#2+100 ;
G01 X#1 F300 ;
```

Explanation

- Variable representation

When specifying a variable, specify a number sign (#) followed by a variable number.

```
#i (i = 1, 2, 3, 4, .....)
[Example]    #5
             #109
             #1005
```

A variable can also be represented as follows using <expression> described in the section about arithmetic and logic operation commands.

```
#[<expression>]
[Example]    #[#100]
             #[#1001-1]
             #[#6/2]
```

Variable #i shown in the following can be replaced with a variable of #[<expression>].

- Types of variables

Variables can be classified as local variables, common variables, and system variables according to the variable number. Each of those variables has its own usage and characteristics. Read-only system constants are also provided.

- Range of variable values

Local and common variables can have a value in the following ranges. If the result of calculation turns out to be invalid, an alarm PS0111 is issued.

When parameter F16 (No.6008#0) = 0

Maximum value: approx. $\pm 10^{308}$
Minimum value: approx. $\pm 10^{-308}$
Numeric data handled by a custom macro conforms to the IEEE standard and is handled as a double-precision real number. An error resulting from operation depends on the precision.

When parameter F16 (No.6008#0) = 1

Maximum value: approx. $\pm 10^{47}$
Minimum value: approx. $\pm 10^{-29}$

- Local variable (#1-#33)

A local variable is a variable that is used in a macro locally. That is, local variable #i used by a macro called at a certain time is different from that used by a macro called at another time, regardless of whether the two macros are the same. Therefore, for example, when macro A calls macro B during multiple calls or the like, it is impossible for macro B to corrupt a local variable used by macro A by erroneously using the variable.

A local variable is used to pass arguments. For information on correspondence between arguments and addresses, see the section about macro calling commands. The initial state of a local variable to which no arguments are passed is <null> and the user can freely use the variable. The attribute of a local variable is READ/WRITE enabled.

- Common variable (#100-#199, #500-#999)

A common variable is shared among the main program, subprograms called by the main program, and macros while a local variable is used locally in a macro. That is, #i used by a macro is the same as that used by another macro. Therefore, a resultant common variable obtained by using a macro can be used by another macro. The attribute of a common variable is basically READ/WRITE enabled. However, the common variable can be protected (its attribute is set to READ only) by specifying its variable number using parameters No.6031 and No.6032. A common variable can be freely used by the user even when its usage is not defined by the system. The number of common variables can be specified by selecting one of the following options.

(a) 100 common variables (specified only with the custom macro option)

The common variables #100 to #149 and #500 to #549 can be used. The variables #100 to #149 are cleared during power-down, but the variables #500 to #549 are retained during power-down.

(b) 600 common variables (specified with the custom macro option or custom macro common variable addition option)

The common variables #100 to #199 and #500 to #999 can be used. The variables #100 to #199 are cleared during power-down, but the variables #500 to #999 are retained during power-down.

- Write protection of a common variable

Multiple common variables (#500 to #999) can be protected (their attributes are set to READ only) by setting variable numbers in parameters No.6031 and No.6032. This protection is enabled for both Input/All Clear by MDI on the macro screen and write operation by a macro program. If the NC program specifies WRITE operation (used in the left side) for a common program in the set range, an alarm PS0116 is issued.

- System variable

A variable whose usage does not vary in the system. The attribute of a system variable is READ only, WRITE only, or READ/WRITE enabled depending on the nature of a system variable.

- System constant

A system constant can be referenced as with a variable even though its value is fixed. The attribute of a system constant is READ only.

- Omission of the decimal point

When a variable value is defined in a program, the decimal point can be omitted.

[Example]

When #1 = 123; is defined, the actual value of variable #1 is 123.000.

- Referencing variables

The value following an address can be replaced with a variable. When programming as <address>#i or <address>-#i, the variable value or the complement of it is used as the specified value of the address.

[Example] F#33 is the same as F1.5 when #33 = 1.5.
Z-#18 is the same as Z-20.0 when #18 = 20.0.
G#130 is the same as G3 when #130 = 3.0.

A variable cannot be referenced using address/, :, or O and N.

[Example] Programming such as O#27 or N#1 is not allowed.
n (n = 1 to 9) in the optional block skip /n cannot be a variable.

A variable number cannot be specified by a direct variable.

[Example] When replacing 5 in #5 with #30, specify #[#30] instead of ##30.
No values exceeding the maximum allowable value for each address can be specified.

[Example] When #140 = 120, G#140 exceeds the maximum allowable value.
When a variable is used as address data, the variable is automatically rounded off to the number of significant figures of each address or less.

[Example] For a machine with an increment system of 1/1000 mm (IS-B), when #1 = 12.3456, G00 X#1; becomes G00 X12.346;.

If <expression>, described later, is used, the value following an address can be replaced with <expression>.

<address>[<expression>] or <address>-[<expression>]

The program code shown above indicates the value of <expression> or the complement of the value is used as an address value. Note that a constant with no decimal point, enclosed in brackets ([ ]), is assumed to have a decimal point at the end.

[Example] X[#24+#18*COS[#1]]
Z-[#18+#26]

- Undefined variable

When the value of a variable is not defined, such a variable is referred to as a "null" variable. Variables #0 and #3100 are always null variables. They cannot be written to, but they can be read.

(a) Quotation
When an undefined variable is quotated, the address itself is also ignored.

Original command	G90 X100 Y#1
Equivalent command when #1 = <null>	G90 X100
Equivalent command when #1 = 0	G90 X100 Y0

(b) Definition/replacement, addition, multiplication
When a local variable or common variable is directly replaced with <null>, the result is <null>. When a system variable is directly replaced with <null> or the result of calculation including <null> is replaced, a variable value of 0 is assumed.

Original expression (local variable)	#2=#1	#2=#1*5	#2=#1+#1
Replacement result (when #1 = <null>)	<null>	0	0
Replacement result (when #1 = 0)	0	0	0

Original expression (common variable)	#100=#1	#100=#1*5	#100=#1+#1
Replacement result (when #1 = <null>)	<null>	0	0
Replacement result (when #1 = 0)	0	0	0

Original expression (system variable)	#2001=#1	#2001=#1*5	#2001=#1+#1
Replacement result (when #1 = <null>)	0	0	0
Replacement result (when #1 = 0)	0	0	0

(c) Comparison
<null> differs from 0 only for EQ and NE.
<null> is equal to 0 for GE, GT, LE, and LT.

- When <null> is assigned to #1

Conditional expression	#1 EQ #0	#1 NE 0	#1 GE #0	#1 GT 0	#1 LE #0	#1 LT 0
Evaluation result	Established (true)	Established (true)	Established (true)	Not established (false)	Established (true)	Not established (false)

- When 0 is assigned to #1

Conditional expression	#1 EQ #0	#1 NE 0	#1 GE #0	#1 GT 0	#1 LE #0	#1 LT 0
Evaluation result	Not established (false)	Not established (false)	Established (true)	Not established (false)	Established (true)	Not established (false)

- Specifying a system variable (constant) by its name

A system variable (constant) is specified by its variable number, but it can also be specified by its predetermined system variable (constant) name. A system variable (constant) name begins with an underscore (_), followed by up to seven uppercase letters, numerics, or underscores. For axis-dependent variables (such as coordinates) or variables having a lot of data of similar types (such as tool compensation), subscript [n] (n: integer) can be used to specify values. In this case, n can be specified in <expression> format (calculation format). The command format must be specified in [#system-variable-name] format, as shown below.

[#_DATE]

[Example]

[#_DATE]=20040117 ;	: 2004.01.17 is assigned to #3011 (year month date).
[#_TIME]=161705 ;	: 16:17:05 is assigned to #3012 (time minute second).
#101=[#_ABSMT[1]] ;	: #5021 (machine coordinate value of the 1st axis) is read off and assigned to #101.
#102=[#_ABSKP[#500*2]] ;	: #506x (skip position of [#500*2]th axis) is read off and assigned to #102.

If a value other than an integer is specified for subscript n, a variable value is referenced, assuming that the fractional portion is rounded off.

[Example]

[#_ABSIO[1.4999999]] :	This value is assumed to be [#_ABSIO[1]], that is, #5001.
[#_ABSIO[1.5000000]] :	This value is assumed to be [#_ABSIO[2]], that is, #5002.

NOTE

1 When the specified variable name is not registered, an alarm PS1098 is issued.

2 When a negative or other invalid subscript is specified, an alarm PS1099 is issued.

- System constant #0, #3100-#3102 (Attribute: R)

Constants used as fixed values in the system can be used as system variables. Such constants are called system constants. The system constants provided are shown below.

Constant number	Constant name	Description
#0, #3100	[#_EMPTY]	Null
#3101	[#_PI]	Circular constant π = 3.14159265358979323846
#3102	[#_E]	Base of natural logarithm e = 2.71828182845904523536

- Specifying a common variable by its name

Specifying a variable name set by the SETVN command described later allows reading from or writing to a common variable.
The command must be specified in the form [#common-variable-name] such as [#VAR500].

[Example]

X[#POS1] Y[#POS2] ;	:	Specifying a position by the variable name
[#POS1] = #100+#101 ;	:	Executing a assignment statement by the variable name
#[100+[#ABS]] = 500 ;	:	Same as above (by a variable number)
#500 = [1000+[#POS2]*10] ;	:	Reading a variable by a variable name

- Setting and specifying the name of a common variable (SETVN)

For the 50 common variables, #500 to #549, a name of up to eight characters can be specified by using a command as shown below.

SETVN n [VAR500, VAR501, VAR502,......] ;

n represents the starting number of a common variable for which the name is specified.
VAR500 is the variable name of variable n, VAR501 is the variable name of variable n+1, and VAR502 is the variable name of variable number n+2, and so on. Each string is delimited by a comma (,). All codes that can be used as meaningful information in a program except control in, control out, [,], EOB, EOR, and : (colon in a program number) can be used. However, each name must begin with an alphabetical character. Variable names are not cleared on switch-off.
Specifying a set variable name allows reading from or writing to the common variable. The command must be specified in the form [#common-variable-name] such as [#VAR500].

[Example] SETVN 510[TOOL_NO, WORK_NO, COUNTER1, COUNTER2];

The command above names the variables as follows.

Variable	Name
#510	TOOL_NO
#511	WORK_NO
#512	COUNTER1
#513	COUNTER2

The names specified by the command can be used in a program. For example, when 10 is assigned to #510, the expression [#TOOL_NO]=10; can be used instead #510=10;.

NOTE

If the same name was specified for different common variables, only the variable which has the smaller variable number can be referenced with the specified name.

16.2 SYSTEM VARIABLES

System variables can be used to read and write internal CNC data such as tool compensation values and current position data. System variables are essential for automation and general-purpose program development.

List of system variables and constants

n represents a subscript.
R, W, and R/W are attributes of a variable and indicate read-only, write-only, and read/write enabled, respectively.

- Interface signals

System variable number	System variable name	Attribute	Description
#1000-#1031	[#_UI[n]]	R	Interface input signals (BIT), UI000-UI031 NOTE) Subscript n represents a BIT position (0-31).
#1032-#1035	[#_UIL[n]]	R	Interface input signals (LONG), UI000-UI031/ UI100-UI131/ UI200-UI231/UI300-UI331 NOTE) Subscript n (0-3): 0 = UI000-UI031, 1 = UI100-UI131, 2 = UI200-231, 3 = UI300-UI331
#1100-#1131	[#_UO[n]]	R/W	Interface output signals (BIT), UO000-UO031 NOTE) Subscript n represents a BIT position (0-31).
#1132-#1135	[#_UOL[n]]	R/W	Interface output signals (LONG), UO000-UO031/ UO100-UO131/UO200-UO231/UO300-UO331 NOTE) Subscript n (0-3): 0 = UO000-UO031, 1 = UO100-UO131, 2 = UO200-231, 3 = UO300-UO331

- Tool compensation value

M

Tool compensation memory A

System variable number	System variable name	Attribute	Description
#2001-#2200 #10001-#10999	[#_OFS[n]]	R/W	Tool compensation value Note) Subscript n represents a compensation number (1 to 200). When the number of sets is larger than 200, the numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory B when parameter V15 (No.6000#3) = 0

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSW[n]]	R/W	Tool compensation value (wear) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSG[n]]	R/W	Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory B when parameter V15 (No.6000#3) = 1

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSG[n]]	R/W	Tool compensation value (geometry) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSW[n]]	R/W	Tool compensation value (H code, wear) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory C when parameter V15 (No.6000#3) = 0

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHW[n]]	R/W	Tool compensation value (H code, wear) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSHG[n]]	R/W	Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#12001-#12999	[#_OFSDW[n]]	R/W	Tool compensation value (D code, wear) Note) Subscript n represents a compensation number (1 to 999).
# 13001-#13999	[#_OFSDG[n]]	R/W	Tool compensation (D code, geometry) Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory C when parameter V15 (No.6000#3) = 1

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHG[n]]	R/W	Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSHW[n]]	R/W	Tool compensation value (H code, wear) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2401-#2600	[#_OFSDG[n]]	R/W	Tool compensation value (D code, geometry) (Note 2) Note 1) Subscript n represents a compensation number (1 to 200). Note 2) Enabled when parameter D15 (No.6004#5) = 1.
#12001-#12999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2601-#2800	[#_OFSDW[n]]	R/W	Tool compensation value (D code, wear) (Note 2) Note 1) Subscript n represents a compensation number (1 to 200). Note 2) Enabled when parameter D15 (No.6004#5) = 1.
#13001-#13999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

- Tool compensation value

T

Without tool geometry/wear compensation memory

System variable number	System variable name	Attribute	Description
#2001-#2064 #10001-#10999	[#_OFSX[n]]	R/W	X-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2101-#2164 #11001-#11999	[#_OFSZ[n]]	R/W	Z-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2264 #12001-#12999	[#_OFSR[n]]	R/W	Tool nose radius compensation value Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2301-#2364 #13001-#13999	[#_OFST[n]]	R/W	Virtual tool tip T position Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2401-#2449 #14001-#14999	[#_OFSY[n]]	R/W	Y-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

With tool geometry/wear compensation memory

System variable number	System variable name	Attribute	Description
#2001-#2064 #10001-#10999	[#_OFSXW[n]]	R/W	X-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2101-#2164 #11001-#11999	[#_OFSZW[n]]	R/W	Z-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2264 #12001-#12999	[#_OFSRW[n]]	R/W	Tool nose radius compensation value (wear) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2301-#2364 #13001-#13999	[#_OFST[n]]	R/W	Virtual tool tip T position Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2401-#2449 #14001-#14999	[#_OFSYW[n]]	R/W	Y-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2451-#2499 #19001-#19999	[#_OFSYG[n]]	R/W	Y-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2701-#2749 #15001-#15999	[#_OFSXG[n]]	R/W	X-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 49).

System variable number	System variable name	Attribute	Description
#2801-#2849 #16001-#16999	[#_OFSZG[n]]	R/W	Z-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2901-#2964 #17001-#17999	[#_OFSRG[n]]	R/W	Tool nose radius compensation value (geometry) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Workpiece coordinate system shift amount

T

System variable number	System variable name	Attribute	Description
#2501	[#_WZ_SFTX]	R/W	X-axis workpiece shift amount
#2601	[#_WZ_SHTZ]	R/W	Z-axis workpiece shift amount

X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes

- Automatic operation or the like

System variable number	System variable name	Attribute	Description
#3000	[#_ALM]	W	Macro alarm
#3001	[#_CLOCK1]	R/W	Clock 1 (ms)
#3002	[#_CLOCK2]	R/W	Clock 2 (hr)
#3003	[#_CNTL1]	R/W	Enable or disable the suppression of single block stop. Enable or disable the waiting of the auxiliary function completion signal.
#3003 bit 0	[#_M_SBK]	R/W	Enable or disable the suppression of single block stop.
#3003 bit 1	[#_M_FIN]	R/W	Enable or disable waiting for the auxiliary function completion signal.
#3004	[#_CNTL2]	R/W	Enable or disable feed hold. Enable or disable feedrate override. Enable or disable exact stop check.
#3004 bit 0	[#_M_FHD]	R/W	Enable or disable feed hold.
#3004 bit 1	[#_M_OV]	R/W	Enable or disable feedrate override.
#3004 bit 2	[#_M_EST]	R/W	Enable or disable exact stop check.
#3005	[#_SETDT]	R/W	Read/write setting data.
#3006	[#_MSGSTP]	W	Stop with a message.
#3007	[#_MRIMG]	R	Status of a mirror image (DI and setting)
#3008	[#_PRSTR]	R	Restarting/not restarting a program

- Time

System variable number	System variable name	Attribute	Description
#3011	[#_DATE]	R	Year/Month/Date
#3012	[#_TIME]	R	Hour/Minute/Second

- Number of parts

System variable number	System variable name	Attribute	Description
#3901	[#_PRTSA]	R/W	Total number of parts
#3902	[#_PRTSN]	R/W	Number of required parts

- Tool compensation memory

M

System variable number	System variable name	Attribute	Description
#3980	[#_OFSMEM]	R	Tool compensation memory information

- Main program number

System variable number	System variable name	Attribute	Description
#4000	[#_MAINO]	R	Main program number

- Modal information

M

System variable number	System variable name	Attribute	Description
#4001-#4030	[#_BUFG[n]]	R	Modal information on blocks that have been specified by last minute (G code) Note) Subscript n represents a G code group number.
#4102	[#_BUFB]	R	Modal information on blocks that have been specified by last minute (B code)
#4107	[#_BUFD]	R	Modal information on blocks that have been specified by last minute (D code)
#4108	[#_BUFE]	R	Modal information on blocks that have been specified by last minute (E code)
#4109	[#_BUFF]	R	Modal information on blocks that have been specified by last minute (F code)
#4111	[#_BUFH]	R	Modal information on blocks that have been specified by last minute (H code)
#4113	[#_BUFM]	R	Modal information on blocks that have been specified by last minute (M code)
#4114	[#_BUFN]	R	Modal information on blocks that have been specified by last minute (sequence number)
#4115	[#_BUFO]	R	Modal information on blocks that have been specified by last minute (program number)
#4119	[#_BUFS]	R	Modal information on blocks that have been specified by last minute (S code)
#4120	[#_BUFT]	R	Modal information on blocks that have been specified by last minute (T code)
#4130	[#_BUFWZP]	R	Modal information on blocks that have been specified by last minute (additional workpiece coordinate system number)
#4201-#4230	[#_ACTG[n]]	R	Modal information on the block currently being executed (G code) Note) Subscript n represents a G code group number.
#4302	[#_ACTB]	R	Modal information on the block currently being executed (B code)
#4307	[#_ACTD]	R	Modal information on the block currently being executed (D code)
#4308	[#_ACTE]	R	Modal information on the block currently being executed (E code)
#4309	[#_ACTF]	R	Modal information on the block currently being executed (F code)
#4311	[#_ACTH]	R	Modal information on the block currently being executed (H code)
#4313	[#_ACTM]	R	Modal information on the block currently being executed (M code)
#4314	[#_ACTN]	R	Modal information on the block currently being executed (sequence number)
#4315	[#_ACTO]	R	Modal information on the block currently being executed (program number)
#4319	[#_ACTS]	R	Modal information on the block currently being executed (S code)

System variable number	System variable name	Attribute	Description
#4320	[#_ACTT]	R	Modal information on the block currently being executed (T code)
#4330	[#_ACTWZP]	R	Modal information on the block currently being executed (additional workpiece coordinate system number)
#4401-#4430	[#_INTG[n]]	R	Modal information on interrupted blocks (G code) Note) Subscript n represents a G code group number.
#4502	[#_INTB]	R	Modal information on interrupted blocks (B code)
#4507	[#_INTD]	R	Modal information on interrupted blocks (D code)
#4508	[#_INTE]	R	Modal information on interrupted blocks (E code)
#4509	[#_INTF]	R	Modal information on interrupted blocks (F code)
#4511	[#_INTH]	R	Modal information on interrupted blocks (H code)
#4513	[#_INTM]	R	Modal information on interrupted blocks (M code)
#4514	[#_INTN]	R	Modal information on interrupted blocks (sequence number)
#4515	[#_INTO]	R	Modal information on interrupted blocks (program number)
#4519	[#_INTS]	R	Modal information on interrupted blocks (S code)
#4520	[#_INTT]	R	Modal information on interrupted blocks (T code)
#4530	[#_INTWZP]	R	Modal information on interrupted blocks (additional workpiece coordinate system number)

T

System variable number	System variable name	Attribute	Description
#4001-#4030	[#_BUFG[n]]	R	Modal information on blocks that have been specified by last minute (G code) Note) Subscript n represents a G code group number.
#4108	[#_BUFE]	R	Modal information on blocks that have been specified by last minute (E code)
#4109	[#_BUFF]	R	Modal information on blocks that have been specified by last minute (F code)
#4113	[#_BUFM]	R	Modal information on blocks that have been specified by last minute (M code)
#4114	[#_BUFN]	R	Modal information on blocks that have been specified by last minute (sequence number)
#4115	[#_BUFO]	R	Modal information on blocks that have been specified by last minute (program number)
#4119	[#_BUFS]	R	Modal information on blocks that have been specified by last minute (S code)

System variable number	System variable name	Attribute	Description
#4120	[#_BUFT]	R	Modal information on blocks that have been specified by last minute (T code)
#4130	[#_BUFWZP]	R	Modal information on blocks that have been specified by last minute (additional workpiece coordinate system number)
#4201-#4230	[#_ACTG[n]]	R	Modal information on the block currently being executed (G code) Note) Subscript n represents a G code group number.
#4308	[#_ACTE]	R	Modal information on the block currently being executed (E code)
#4309	[#_ACTF]	R	Modal information on the block currently being executed (F code)
#4313	[#_ACTM]	R	Modal information on the block currently being executed (M code)
#4314	[#_ACTN]	R	Modal information on the block currently being executed (sequence number)
#4315	[#_ACTO]	R	Modal information on the block currently being executed (program number)
#4319	[#_ACTS]	R	Modal information on the block currently being executed (S code)
#4320	[#_ACTT]	R	Modal information on the block currently being executed (T code)
#4330	[#_ACTWZP]	R	Modal information on the block currently being executed (additional workpiece coordinate system number)
#4401-#4430	[#_INTG[n]]	R	Modal information on interrupted blocks (G code) Note) Subscript n represents a G code group number.
#4508	[#_INTE]	R	Modal information on interrupted blocks (E code)
#4509	[#_INTF]	R	Modal information on interrupted blocks (F code)
#4513	[#_INTM]	R	Modal information on interrupted blocks (M code)
#4514	[#_INTN]	R	Modal information on interrupted blocks (sequence number)
#4515	[#_INTO]	R	Modal information on interrupted blocks (program number)
#4519	[#_INTS]	R	Modal information on interrupted blocks (S code)
#4520	[#_INTT]	R	Modal information on interrupted blocks (T code)
#4530	[#_INTWZP]	R	Modal information on interrupted blocks (additional workpiece coordinate system number)

- Position information

System variable number	System variable name	Attribute	Description
#5001-#5020	[#_ABSIO[n]]	R	End point position of the previous block (workpiece coordinate system) Note) Subscript n represents a axis number (1 to 20)
#100001-#100050			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5021-#5040	[#_ABSMT[n]]	R	Specified current position (machine coordinate system) Note) Subscript n represents a axis number (1 to 20).
#100051-#100100			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5041-#5060	[#_ABSOT[n]]	R	Specified current position (workpiece coordinate system) Note) Subscript n represents a axis number (1 to 20).
#100101-#100150			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5061-#5080	[#_ABSKP[n]]	R	Skip position (workpiece coordinate system) Note) Subscript n represents a axis number (1 to 20).
#100151-#100200			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).

- Tool length compensation value

M

System variable number	System variable name	Attribute	Description
#5081-#5100	[#_TOFS[n]]	R	Tool length compensation value Note) Subscript n represents a axis number (1 to 20).
#100201-#100250			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).

- Tool offset value

T

System variable number	System variable name	Attribute	Description
#5081	[#_TOFSWX]	R	X-axis tool offset (wear)
#508 2	[#_TOFSWZ]		Y-axis tool offset (wear)
#5083	[#_TOFSWY]		Z-axis tool offset (wear)
#5121	[#_TOFSGX]	R	X-axis tool offset (geometry)
#5122	[#_TOFSGZ]		Y-axis tool offset (geometry)
#5123	[#_TOFSGY]		Z-axis tool offset (geometry)

X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Servo position deviation

System variable number	System variable name	Attribute	Description
#5101-#5120	[#_SVERR[n]]	R	Servo positional deviation Note) Subscript n represents a axis number (1 to 20).
#100251-#100300			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).

- Manual handle interruption

System variable number	System variable name	Attribute	Description
#5121-#5140	[#_MIRTP[n]]	R	Manual handle interruption Note) Subscript n represents a axis number (1 to 20).
#100651-#100700			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).

- Distance to go

System variable number	System variable name	Attribute	Description
#5181-#5200	[#_DIST[n]]	R	Distance to go Note) Subscript n represents a axis number (1 to 20).
#100801-#100850			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).

- Workpiece origin offset value, extended workpiece origin offset value

M

System variable number	System variable name	Attribute	Description
#5201-#5220	[#_WZCMN[n]]	R/W	Common workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5221-#5240	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5241-#5260	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5261-#5280	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5281-#5300	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5301-#5320	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5321-#5340	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#100301-#100350	[#_WZCMN[n]]	R/W	Common workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100351-#100400	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100401-#100450	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100451-#100500	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100501-#100550	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100551-#100600	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100601-#100650	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
Extended workpiece origin offset value			
#7001-#7020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#7021-#7040	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).

System variable number	System variable name	Attribute	Description
: :	: :	: :	: :
#7941-#7960	[#_WZP48[n]]	R/W	G54.1P48 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#14001-#14020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#14051-#14100	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
: :	: :	: :	: :
#19971-#20000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#101001-#101050	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#101051-#101100	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
: :	: :	: :	: :
#115901-#115950	[#_WZP299[n]]	R/W	G54.1P299 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#115951-#116000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).

T

System variable number	System variable name	Attribute	Description
#5201-#5220	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5221-#5240	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5241-#5260	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5261-#5280	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5281-#5300	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#5301-#5320	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).

System variable number	System variable name	Attribute	Description
#5321-#5340	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#100301-#100350	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100351-#100400	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100401-#100450	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100451-#100500	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100501-#100550	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100551-#100600	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#100601-#100650	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
Extended workpiece origin offset value			
#7001-#7020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#7021-#7040	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
: :	: :	: :	: :
#7941-#7960	[#_WZP48[n]]	R/W	G54.1P48 workpiece origin offset value Note) Subscript n represents a axis number (1 to 20).
#101001-#101050	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#101051-#101100	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
: :	: :	: :	: :
#115901-#115950	[#_WZP299[n]]	R/W	G54.1P299 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).
#115951-#116000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents a axis number (1 to 50).

- Skip position (detection unit)

System variable number	System variable name	Attribute	Description
#5421-#5440	[#_SKPDTC[n]]	R	Skip position (detection unit) Note) Subscript n represents a axis number (1 to 20).
#100701-#100750			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).

- Rotary table dynamic fixture offset value

M

System variable number	System variable name	Attribute	Description
#5500	[#_FOFSP]	R	Number of the standard fixture offset being selected (P)
#5501-#5520	[#_FOFSVAL[n]]	R	Offset value of the standard fixture offset being selected Note) Subscript n represents a axis number (1 to 20).
#117001-#117050			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5521-#5540	[#_FOFS1[n]]	R/W	Standard fixture offset value (first set) Note) Subscript n represents a axis number (1 to 20).
#117051-#117100			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5541-#5560	[#_FOFS2[n]]	R/W	Standard fixture offset value (second set) Note) Subscript n represents a axis number (1 to 20).
#117101-#117150			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5561-#5580	[#_FOFS3[n]]	R/W	Standard fixture offset value (third set) Note) Subscript n represents a axis number (1 to 20).
#117151-#117200			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5581-#5600	[#_FOFS4[n]]	R/W	Standard fixture offset value (fourth set) Note) Subscript n represents a axis number (1 to 20).
#117201-#117250			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).

System variable number	System variable name	Attribute	Description
#5601-#5620	[#_FOFS5[n]]	R/W	Standard fixture offset value (fifth set) Note) Subscript n represents a axis number (1 to 20).
#117251-#117300			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5621-#5640	[#_FOFS6[n]]	R/W	Standard fixture offset value (sixth set) Note) Subscript n represents a axis number (1 to 20).
#117301-#117350			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5641-#5660	[#_FOFS7[n]]	R/W	Standard fixture offset value (seventh set) Note) Subscript n represents a axis number (1 to 20).
#117351-#117400			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).
#5661-#5680	[#_FOFS8[n]]	R/W	Standard fixture offset value (eighth set) Note) Subscript n represents a axis number (1 to 20).
#117401-#117450			The numbers to the left can also be used. Note) Subscript n represents a axis number (1 to 50).

- Dynamic standard tool compensation value

M

System variable number	System variable name	Attribute	Description
#118051-#118100	[#_DOFS1[n]]	R/W	Dynamic standard tool compensation value (first set) Note) Subscript n represents a axis number (1 to 50).
#118101-#118150	[#_DOFS2[n]]	R/W	Dynamic standard tool compensation value (second set) Note) Subscript n represents a axis number (1 to 50).
#118151-#118200	[#_DOFS3[n]]	R/W	Dynamic standard tool compensation value (third set) Note) Subscript n represents a axis number (1 to 50).
#118201-#118250	[#_DOFS4[n]]	R/W	Dynamic standard tool compensation value (fourth set) Note) Subscript n represents a axis number (1 to 50).
#118251-#118300	[#_DOFS5[n]]	R/W	Dynamic standard tool compensation value (fifth set) Note) Subscript n represents a axis number (1 to 50).
#118301-#118350	[#_DOFS6[n]]	R/W	Dynamic standard tool compensation value (sixth set) Note) Subscript n represents a axis number (1 to 50).
#118351-#118400	[#_DOFS7[n]]	R/W	Dynamic standard tool compensation value (seventh set) Note) Subscript n represents a axis number (1 to 50).
#118401-#118450	[#_DOFS8[n]]	R/W	Dynamic standard tool compensation value (eighth set) Note) Subscript n represents a axis number (1 to 50).

- System constant

System constant number	System constant name	Attribute	Description
#0, #3100	[#_EMPTY]	R	Null
#3101	[#_PI]	R	Circular constant π = 3.14159265358979323846
#3102	[#_E]	R	Base of natural logarithm e = 2.71828182845904523536

Explanation

R, W, and R/W are attributes of a variable and represents read-only, write-only, and read/write enabled, respectively.

- Interface signal #1000-#1031, #1032, #1033-#1035 (Attribute: R) #1100-#1115, #1132, #1133-#1135 (Attribute: R/W)

[Input signal]
The status of interface input signals can be obtained by reading the value of system variables #1000 to #1032.

Variable number	Variable name	Point	Interface input signal
#1000	[#_UI[0]]	1	UI000 (2^0)
#1001	[#_UI[1]]	1	UI001 (2^1)
#1002	[#_UI[2]]	1	UI002 (2^2)
#1003	[#_UI[3]]	1	UI003 (2^3)
#1004	[#_UI[4]]	1	UI004 (2^4)
#1005	[#_UI[5]]	1	UI005 (2^5)
#1006	[#_UI[6]]	1	UI006 (2^6)
#1007	[#_UI[7]]	1	UI007 (2^7)
#1008	[#_UI[8]]	1	UI008 (2^8)
#1009	[#_UI[9]]	1	UI009 (2^9)
#1010	[#_UI[10]]	1	UI010 (2^{10})
#1011	[#_UI[11]]	1	UI011 (2^{11})
#1012	[#_UI[12]]	1	UI012 (2^{12})
#1013	[#_UI[13]]	1	UI013 (2^{13})
#1014	[#_UI[14]]	1	UI014 (2^{14})
#1015	[#_UI[15]]	1	UI015 (2^{15})
#1016	[#_UI[16]]	1	UI016 (2^{16})
#1017	[#_UI[17]]	1	UI017 (2^{17})
#1018	[#_UI[18]]	1	UI018 (2^{18})
#1019	[#_UI[19]]	1	UI019 (2^{19})
#1020	[#_UI[20]]	1	UI020 (2^{20})
#1021	[#_UI[21]]	1	UI021 (2^{21})
#1022	[#_UI[22]]	1	UI022 (2^{22})
#1023	[#_UI[23]]	1	UI023 (2^{23})
#1024	[#_UI[24]]	1	UI024 (2^{24})
#1025	[#_UI[25]]	1	UI025 (2^{25})
#1026	[#_UI[26]]	1	UI026 (2^{26})
#1027	[#_UI[27]]	1	UI027 (2^{27})
#1028	[#_UI[28]]	1	UI028 (2^{28})
#1029	[#_UI[29]]	1	UI029 (2^{29})
#1030	[#_UI[30]]	1	UI030 (2^{30})
#1031	[#_UI[31]]	1	UI031 (2^{31})
#1032	[#_UIL[0]]	32	UI000-UI031
#1033	[#_UIL[1]]	32	UI100-UI131
#1034	[#_UIL[2]]	32	UI200-UI231
#1035	[#_UIL[3]]	32	UI300-UI331

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

Since the read value is 1.0 or 0.0 regardless of the unit system, the unit system must be considered when a macro is created.
The input signals at 32 points can be read at a time by reading from system variables #1032 to #1035.

$$\#1032 = \sum_{i=0}^{30} \#[1000+i] \times 2^i - \#1031 \times 2^{31}$$

$$\#[1032+n] = \sum_{i=0}^{30} \{2^i \times V_i\} - 2^{31} \times V_{31}$$

When $UIn_i = 0$, $V_i = 0$.
When $UIn_i = 1$, $V_i = 1$.
n = 0-3

[Output signal]
Interface output signals can be sent by assigning values to system variables #1100 to #1132 for sending interface signals.

Variable number	Variable name	Point	Interface input signal
#1100	[#_UO[0]]	1	UO000 (2^{0})
#1101	[#_UO[1]]	1	UO001 (2^{1})
#1102	[#_UO[2]]	1	UO002 (2^{2})
#1103	[#_UO[3]]	1	UO003 (2^{3})
#1104	[#_UO[4]]	1	UO004 (2^{4})
#1105	[#_UO[5]]	1	UO005 (2^{5})
#1106	[#_UO[6]]	1	UO006 (2^{6})
#1107	[#_UO[7]]	1	UO007 (2^{7})
#1108	[#_UO[8]]	1	UO008 (2^{8})
#1109	[#_UO[9]]	1	UO009 (2^{9})
#1110	[#_UO[10]]	1	UO010 (2^{10})
#1111	[#_UO[11]]	1	UO011 (2^{11})
#1112	[#_UO[12]]	1	UO012 (2^{12})
#1113	[#_UO[13]]	1	UO013 (2^{13})
#1114	[#_UO[14]]	1	UO014 (2^{14})
#1115	[#_UO[15]]	1	UO015 (2^{15})
#1116	[#_UO[16]]	1	UO016 (2^{16})
#1117	[#_UO[17]]	1	UO017 (2^{17})
#1118	[#_UO[18]]	1	UO018 (2^{18})
#1119	[#_UO[19]]	1	UO019 (2^{19})
#1120	[#_UO[20]]	1	UO020 (2^{20})
#1121	[#_UO[21]]	1	UO021 (2^{21})
#1122	[#_UO[22]]	1	UO022 (2^{22})
#1123	[#_UO[23]]	1	UO023 (2^{23})
#1124	[#_UO[24]]	1	UO024 (2^{24})
#1125	[#_UO[25]]	1	UO025 (2^{25})

Variable number	Variable name	Point	Interface input signal
#1126	[#_UO[26]]	1	UO026 (2^{26})
#1127	[#_UO[27]]	1	UO027 (2^{27})
#1128	[#_UO[28]]	1	UO028 (2^{28})
#1129	[#_UO[29]]	1	UO029 (2^{29})
#1130	[#_UO[30]]	1	UO030 (2^{30})
#1131	[#_UO[31]]	1	UO031 (2^{31})
#1132	[#_UOL[0]]	32	UO000-UO031
#1133	[#_UOL[1]]	32	UO100-UO131
#1134	[#_UOL[2]]	32	UO200-UO231
#1135	[#_UOL[3]]	32	UO300-UO331

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

The output signals at 32 points can be written at a time by writing to system variables #1132 to #1135. The signals can also be read.

$$\#1132 = \sum_{i=0}^{30} \#[1000+i] \times 2^{i} - \#1131 \times 2^{31}$$

$$\#[1132+n] = \sum_{i=0}^{30} \left\{2^{i} \times V_{i}\right\} - 2^{31} \times V_{31}$$

When UIn_i = 0, V_i = 0.
When UIn_i = 1, V_i = 1.
n = 0-3

NOTE

1 When a value other than 1.0 or 0.0 is assigned to variables #1100 to #1131, it is assumed as follows.
<null> is assumed to be 0.
A value other than <null> or 0 is assumed to be 1.
Where, a value less than 0.00000001 is undefined.

2 When any of UI016 to UI031, UI100 to UI131, UI200 to UI231, UI300 to UI331, UO016 to UO031, UO200 to UO231, and UO300 to UO331 are used, parameter MIF (No.6001#0) must be set to 1.

Example

Structure of DI

2^{15}	2^{14}	2^{13}	2^{12}	2^{11}	2^{10}	2^{9}	2^{8}	2^{7}	2^{6}	2^{5}	2^{4}	2^{3}	2^{2}	2^{1}	2^{0}
Used for other purposes			Sign	10^2				10^1				10^0			

Structure of DO

	2^{8}	2^{7}	2^{6}	2^{5}	2^{4}	2^{3}	2^{2}	2^{1}	2^{0}
Not used	Used for other purposes					Address			

<1> Address switching signed BCD 3 digits are read.

Macro calling instruction

```
G65 P9100 D (address);
```

A custom macro body is created as follows.

```
O9100 ;
#1132 = #1132 AND 496 OR#7 ;   :  Address sending
G65 P9101 T60 ;                :  Timer macro
#100 = BIN[#1032 AND 4095] ;   :  BCD 3 digits are read.
IF [#1012 EQ   0] GOTO   9100 ; :  A sign is attached.
#100 = -#100
N9100 M99
```

<2> Eight types of address switching signed BCD 6 digits (3-digit integer part + 3-digit fractional part) are read into #101.

Structure on the machine side

When DO 2^0 = 0: Data with 3 decimal places
When DO 2^0 = 1: Data with 3-digit integer part
When DO 2^3 to 2^1 = 000: No1 data when #1 = 0
When DO 2^3 to 2^1 = 001: No2 data when #2 = 0
:
When DO 2^3 to 2^1 = 111: No8 data when #8 = 0

Macro calling instruction

```
G65    P9101    D (data number);
```

A custom macro body is created as follows.

```
O9101 ;
G65 P9101 D[#1*2+1] ;
#101 = #100   ;
G65 P9100 D[#1*2] ;
#101 = #101 + #100/1000 ;
M99 ;
```

- Tool compensation value #2001-#2800, #10001-#13999 (Attribute: R/W)

M

The compensation values can be obtained by reading system variables #2001 to #2800 or #10001 to #13999 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

<1> Tool compensation memory A

- When the number of compensations is 200 or less

Compensation number	Variable number	Variable name
1	#2001	[#_OFS[1]]
2	#2002	[#_OFS[2]]
3	#2003	[#_OFS[3]]
:	:	:
199	#2199	[#_OFS[199]]
200	#2200	[#_OFS[200]]

- When the number of compensations is more than 200 (For compensation with a compensation number of 200 or less, #2001 to #2200 can also be used.)

Compensation number	Variable number	Variable name
1	#10001	[#_OFS[1]]
2	#10002	[#_OFS[2]]
3	#10003	[#_OFS[3]]
:	:	:
998	#10998	[#_OFS[998]]
999	#10999	[#_OFS[999]]

<2> Tool compensation memory B

- When the number of compensations is 200 or less
  When parameter V15 (No.6000#3) = 0

Compensation number	Wear		Geometry	
	Variable number	Variable name	Variable number	Variable name
1	#2001	[#_OFSW[1]]	#2201	[#_OFSG[1]]
2	#2002	[#_OFSW[2]]	#2202	[#_OFSG[2]]
3	#2003	[#_OFSW[3]]	#2203	[#_OFSG[3]]
:	:	:	:	:
199	#2199	[#_OFSW[199]]	#2399	[#_OFSG[199]]
200	#2200	[#_OFSW[200]]	#2400	[#_OFSG[200]]

When parameter V15 (No.6000#3) = 1

Compensation number	Wear		Geometry	
	Variable number	Variable name	Variable number	Variable name
1	#2201	[#_OFSW[1]]	#2001	[#_OFSG[1]]
2	#2202	[#_OFSW[2]]	#2002	[#_OFSG[2]]
3	#2203	[#_OFSW[3]]	#2003	[#_OFSG[3]]
:	:	:	:	:
199	#2399	[#_OFSW[199]]	#2199	[#_OFSG[199]]
200	#2400	[#_OFSW[200]]	#2200	[#_OFSG[200]]

- When the number of compensations is more than 200 (For compensation with a compensation number of 200 or less, #2001 to #2400 can also be used.)
  When parameter V15 (No.6000#3) = 0

Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#11001	[#_OFSG[1]]	#10001	[#_OFSW[1]]
2	#11002	[#_OFSG[2]]	#10002	[#_OFSW[2]]
3	#11003	[#_OFSG[3]]	#10003	[#_OFSW[3]]
:	:	:	:	:
998	#11998	[#_OFSG[998]]	#10998	[#_OFSW[998]]
999	#11999	[#_OFSG[999]]	#10999	[#_OFSW[999]]

When parameter V15 (No.6000#3) = 1

Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#10001	[#_OFSG[1]]	#11001	[#_OFSW[1]]
2	#10002	[#_OFSG[2]]	#11002	[#_OFSW[2]]
3	#10003	[#_OFSG[3]]	#11003	[#_OFSW[3]]
:	:	:	:	:
998	#10998	[#_OFSG[998]]	#11998	[#_OFSW[998]]
999	#10999	[#_OFSG[999]]	#11999	[#_OFSW[999]]

<3> Tool offset memory C

- When the number of compensations is 200 or less
  When parameter V15 (No.6000#3) = 0

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2201	[#_OFSHG[1]]	#2001	[#_OFSHW[1]]
2	#2202	[#_OFSHG[2]]	#2002	[#_OFSHW[2]]
3	#2203	[#_OFSHG[3]]	#2003	[#_OFSHW[3]]
:	:	:	:	:
199	#2399	[#_OFSHG[199]]	#2199	[#_OFSHW[199]]
200	#2400	[#_OFSHG[200]]	#2200	[#_OFSHW[200]]

When parameter V15 (No.6000#3) = 1

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2001	[#_OFSHG[1]]	#2201	[#_OFSHW[1]]
2	#2002	[#_OFSHG[2]]	#2202	[#_OFSHW[2]]
3	#2003	[#_OFSHG[3]]	#2203	[#_OFSHW[3]]
:	:	:	:	:
199	#2199	[#_OFSHG[199]]	#2399	[#_OFSHW[199]]
200	#2200	[#_OFSHG[200]]	#2400	[#_OFSHW[200]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2401	[#_OFSDG[1]]	#2601	[#_OFSDW[1]]
2	#2402	[#_OFSDG[2]]	#2602	[#_OFSDW[2]]
3	#2403	[#_OFSDG[3]]	#2603	[#_OFSDW[3]]
:	:	:	:	:
199	#2599	[#_OFSDG[199]]	#2799	[#_OFSDW[199]]
200	#2600	[#_OFSDG[200]]	#2800	[#_OFSDW[200]]

NOTE

1 When #2401 to #2800 are used for reading or writing of D codes, parameter D15 (No.6004#5) must be set to 1.

2 When parameter D15 (No.6004#5) is set to 1, system variables #2500 to #2806 for workpiece origin offset cannot be used. Use system variables #5201 to #5324.

- When the number of compensations is more than 200 (For compensation with a compensation number of 200 or less, #2001 to #2800 can also be used.)
  When parameter V15 (No.6000#3) = 0

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#11001	[#_OFSHG[1]]	#10001	[#_OFSHW[1]]
2	#11002	[#_OFSHG[2]]	#10002	[#_OFSHW[2]]
3	#11003	[#_OFSHG[3]]	#10003	[#_OFSHW[3]]
:	:	:	:	:
998	#11998	[#_OFSHG[998]]	#10998	[#_OFSHW[998]]
999	#11999	[#_OFSHG[999]]	#10999	[#_OFSHW[999]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#13001	[#_OFSDG[1]]	#12001	[#_OFSDW[1]]
2	#13002	[#_OFSDG[2]]	#12002	[#_OFSDW[2]]
3	#13003	[#_OFSDG[3]]	#12003	[#_OFSDW[3]]
:	:	:	:	:
998	#13998	[#_OFSDG[998]]	#12998	[#_OFSDW[998]]
999	#13999	[#_OFSDG[999]]	#12999	[#_OFSDW[999]]

When parameter V15 (No.6000#3) = 1

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#10001	[#_OFSHG[1]]	#11001	[#_OFSHW[1]]
2	#10002	[#_OFSHG[2]]	#11002	[#_OFSHW[2]]
3	#10003	[#_OFSHG[3]]	#11003	[#_OFSHW[3]]
:	:	:	:	:
998	#10998	[#_OFSHG[998]]	#11998	[#_OFSHW[998]]
999	#10999	[#_OFSHG[999]]	#11999	[#_OFSHW[999]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#12001	[#_OFSDG[1]]	#13001	[#_OFSDW[1]]
2	#12002	[#_OFSDG[2]]	#13002	[#_OFSDW[2]]
3	#12003	[#_OFSDG[3]]	#13003	[#_OFSDW[3]]
:	:	:	:	
998	#12998	[#_OFSDG[998]]	#13998	[#_OFSDW[998]]
999	#12999	[#_OFSDG[999]]	#13999	[#_OFSDW[999]]

- Tool compensation value #2001-#2964, #10001-#19999 (Attribute: R/W)

T

The compensation values can be obtained by reading system variables #2001 to #2964 or #10001 to #19999 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

<1> Without tool geometry/wear compensation memory

- When the number of compensations is 64 or less

Compensation number	Variable number	Variable name	Description
1	#2001	[#_OFSX[1]]	X-axis compensation value (*1)
2	#2002	[#_OFSX[2]]	
3	#2003	[#_OFSX[3]]	
:	:	:	
63	#2063	[#_OFSX[63]]	
64	#2064	[#_OFSX[64]]	
1	#2101	[#_OFSZ[1]]	Z-axis compensation value (*1)
2	#2102	[#_OFSZ[2]]	
3	#2103	[#_OFSZ[3]]	
:	:	:	
63	#2163	[#_OFSZ[63]]	
64	#2164	[#_OFSZ[64]]	
1	#2201	[#_OFSR[1]]	Tool nose radius compensation value
2	#2202	[#_OFSR[2]]	
3	#2203	[#_OFSR[3]]	
:	:	:	
63	#2263	[#_OFSR[63]]	
64	#2264	[#_OFSR[64]]	
1	#2301	[#_OFST[1]]	Virtual tool tipT position
2	#2302	[#_OFST[2]]	
3	#2303	[#_OFST[3]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSY[1]]	Y-axis compensation value (*1)
2	#2402	[#_OFSY[2]]	
3	#2403	[#_OFSY[3]]	
:	:	:	
48	#2448	[#_OFSY[48]]	
49	#2449	[#_OFSY[49]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- When the number of compensations is more than 64 (For compensation with a compensation number of 64 or less, #2001 to #2449 can also be used.)

Compensation number	Variable number	Variable name	Description
1	#10001	[#_OFSX[1]]	X-axis compensation value (*1)
2	#10002	[#_OFSX[2]]	
3	#10003	[#_OFSX[3]]	
:	:	:	
998	#10998	[#_OFSX[998]]	
999	#10999	[#_OFSX[999]]	
1	#11001	[#_OFSZ[1]]	Z-axis compensation value (*1)
2	#11002	[#_OFSZ[2]]	
3	#11003	[#_OFSZ[3]]	
:	:	:	
998	#11998	[#_OFSZ[998]]	
999	#11999	[#_OFSZ[999]]	
1	#12001	[#_OFSR[1]]	Tool nose radius compensation value
2	#12002	[#_OFSR[2]]	
3	#12003	[#_OFSR[3]]	
:	:	:	
998	#12998	[#_OFSR[998]]	
999	#12999	[#_OFSR[999]]	
1	#13001	[#_OFST[1]]	Virtual tool tip T position
2	#13002	[#_OFST[2]]	
3	#13003	[#_OFST[3]]	
:	:	:	
998	#13998	[#_OFST[998]]	
999	#13999	[#_OFST[999]]	
1	#14001	[#_OFSY[1]]	Y-axis compensation value (*1)
2	#14002	[#_OFSY[2]]	
3	#14003	[#_OFSY[3]]	
:	:	:	
998	#14998	[#_OFSY[998]]	
999	#14999	[#_OFSY[999]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

<2> With tool geometry/wear compensation memory

- When the number of compensations is 64 or less

Compensation number	Variable number	Variable name	Description
1	#2001	[#_OFSXW[1]]	X-axis compensation value (wear) (*1)
2	#2002	[#_OFSXW[2]]	
3	#2003	[#_OFSXW[3]]	
:	:	:	
63	#2063	[#_OFSXW[63]]	
64	#2064	[#_OFSXW[64]]	
1	#2101	[#_OFSZW[1]]	Z-axis compensation value (wear) (*1)
2	#2102	[#_OFSZW[2]]	
3	#2103	[#_OFSZW[3]]	
:	:	:	
63	#2163	[#_OFSZW[63]]	
64	#2164	[#_OFSZW[64]]	
1	#2201	[#_OFSRW[1]]	Tool nose radius compensation value (wear)
2	#2202	[#_OFSRW [2]]	
3	#2203	[#_OFSRW [3]]	
:	:	:	
63	#2263	[#_OFSRW [63]]	
64	#2264	[#_OFSRW [64]]	
1	#2301	[#_OFST[1]]	Virtual tool tip T position
2	#2302	[#_OFST[2]]	
3	#2303	[#_OFST[3]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSYW[1]]	Y-axis compensation value (wear) (*1)
2	#2402	[#_OFSYW [2]]	
3	#2403	[#_OFSYW [3]]	
:	:	:	
48	#2448	[#_OFSYW [48]]	
49	#2449	[#_OFSYW [49]]	
1	#2451	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#2452	[#_OFSYG [2]]	
3	#2453	[#_OFSYG [3]]	
:	:	:	
48	#2498	[#_OFSYG [48]]	
49	#2499	[#_OFSYG [49]]	
1	#2701	[#_OFSXG[1]]	X-axis compensation value (geometry) (*1)
2	#2702	[#_OFSXG[2]]	
3	#2703	[#_OFSXG [3]]	
:	:	:	
48	#2748	[#_OFSXG [48]]	
49	#2749	[#_OFSXG [49]]	
1	#2801	[#_OFSZG[1]]	Z-axis compensation value (geometry) (*1)
2	#2802	[#_OFSZG[2]]	
3	#2803	[#_OFSZG[3]]	
:	:	:	
48	#2848	[#_OFSZG[48]]	
49	#2849	[#_OFSZG[49]]	

Compensation number	Variable number	Variable name	Description
1	#2901	[#_OFSRG[1]]	Tool nose radius compensation value (geometry)
2	#2902	[#_OFSRG[2]]	
3	#2903	[#_OFSRG[3]]	
:	:	:	
63	#2963	[#_OFSRG[63]]	
64	#2964	[#_OFSRG[64]]	
1	#19001	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#19002	[#_OFSYG[2]]	
3	#19003	[#_OFSYG[3]]	
:	:	:	
998	#19998	[#_OFSYG[998]]	
999	#19999	[#_OFSYG[999]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- When the number of compensations is more than 64 (For compensation with a compensation number of 64 or less, #2001 to #2964 or #10001 to #19999 can also be used.)

Compensation number	Variable number	Variable name	Description
1	#10001	[#_OFSXW[1]]	X-axis compensation value (wear) (*1)
2	#10002	[#_OFSXW[2]]	
3	#10003	[#_OFSXW[3]]	
:	:	:	
998	#10998	[#_OFSXW[998]]	
999	#10999	[#_OFSXW[999]]	
1	#11001	[#_OFSZW[1]]	Z-axis compensation value (wear) (*1)
2	#11002	[#_OFSZW[2]]	
3	#11003	[#_OFSZW[3]]	
:	:	:	
998	#11998	[#_OFSZW[998]]	
999	#11999	[#_OFSZW[999]]	
1	#12001	[#_OFSRW[1]]	Tool nose radius compensation value (wear)
2	#12002	[#_OFSRW [2]]	
3	#12003	[#_OFSRW [3]]	
:	:	:	
998	#12998	[#_OFSRW [998]]	
999	#12999	[#_OFSRW [999]]	
1	#13001	[#_OFST[1]]	Virtual tool tip T position
2	#13002	[#_OFST[2]]	
3	#13003	[#_OFST[3]]	
:	:	:	
998	#13998	[#_OFST[998]]	
999	#13999	[#_OFST[999]]	
1	#14001	[#_OFSYW[1]]	Y-axis compensation value (wear) (*1)
2	#14002	[#_OFSYW [2]]	
3	#14003	[#_OFSYW [3]]	
:	:	:	
998	#14998	[#_OFSYW [998]]	
999	#14999	[#_OFSYW [999]]	

Compensation number	Variable number	Variable name	Description
1	#15001	[#_OFSXG[1]]	X-axis compensation value (geometry) (*1)
2	#15002	[#_OFSXG[2]]	
3	#15003	[#_OFSXG [3]]	
:	:	:	
998	#15998	[#_OFSXG [998]]	
999	#15999	[#_OFSXG [999]]	
1	#16001	[#_OFSZG[1]]	Z-axis compensation value (geometry) (*1)
2	#16002	[#_OFSZG[2]]	
3	#16003	[#_OFSZG[3]]	
:	:	:	
998	#16998	[#_OFSZG[998]]	
999	#16999	[#_OFSZG[999]]	
1	#17001	[#_OFSRG[1]]	Tool nose radius compensation value (geometry)
2	#17002	[#_OFSRG[2]]	
3	#17003	[#_OFSRG[3]]	
:	:	:	
998	#17998	[#_OFSRG[998]]	
999	#17999	[#_OFSRG[999]]	
1	#19001	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#19002	[#_OFSYG[2]]	
3	#19003	[#_OFSYG[3]]	
:	:	:	
998	#19998	[#_OFSYG[998]]	
999	#19999	[#_OFSYG[999]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Alarm #3000 (Attribute: W)

When an error is detected in a macro, an unit can enter the alarm state. In addition, an alarm message of up to 26 characters can be specified between a control-out and a control-in after the expression. When an alarm message is not specified, a macro alarm is used instead.

Variable number	Variable name	Description
#3000	[#_ALM]	Macro alarm

When parameter MCA (No.6008#1) = 0

#3000 = n (ALARM MESSAGE); (n: 0-200)

On the screen, the alarm number obtained by adding the value of #3000 to 3000 and alarm message appear.

(Example) #3000 = 1 (ALARM MESSAGE);

→ "3001 ALARM MESSAGE" appears on the alarm screen.

Parameter MCA (No.6008#1) = 1

#3000 = n (ALARM MESSAGE); (n: 0-4095)

On the screen, the alarm number of #3000 and alarm message appear after MC.

(Example) #3000=1 (ALARM MESSAGE);
→ "MC0001 ALARM MESSAGE" appears on the alarm screen.

- Clock #3001, #3002 (Attribute: R/W)

The clock time can be obtained by reading system variables #3001 and #3002 for clocks. The time can be preset by entering a value in the system variables.

Type	Variable number	Variable name	Unit	At power-on	Count condition
Clock 1	#3001	[#_CLOCK1]	1 ms	Reset to 0	Anytime
Clock 2	#3002	[#_CLOCK2]	1 hour	Same as at power-down	When the STL signal is on

The clock accuracy is 16 ms. Clock 1 returns to 0 after a lapse of 2147483648 ms. Clock 2 returns to 0 after a lapse of 9544.37176 hours.

[Example]

```
Timer
Macro calling command
G65 P9101 T (wait time) ms ;
A macro is created as follows.
O9101;
#3001 = 0:   Initial setting
WHILE [#3001 LE #20] DO1:   Wait for a specified time
END1 ;
M99 ;
```

- Controlling of single block stop and waiting for the auxiliary function completion signal #3003 (Attribute: R/W)

Assigning the following values in system variable #3003 allows the specification of whether single block stop is disabled in the following blocks or whether a wait for the completion signal (FIN) of the auxiliary function (M, S, T, or B) before going to the next block is enabled. When a wait for completion signal is disabled, the distribution end signal (DEN) is not sent. Be careful not to specify the next auxiliary function without waiting for the completion signal.

Variable number and variable name	Value	Single block stop	Auxiliary function completion signal
#3003 [#_CNTL1]	0	Enabled	Waiting
	1	Disabled	Waiting
	2	Enabled	Not waiting
	3	Disabled	Not waiting

In addition, the following variable names can be used to enable or disable single block stop and a wait for the auxiliary function completion signal, individually.

Variable name	Value	Single block stop	Auxiliary function completion
[#_M_SBK]	0	Enabled	-
	1	Disabled	-
[#_M_FIN]	0	-	Waiting
	1	-	Not waiting

[Example]

Drill cycle (for incremental programming)
(G81 equivalent)
Macro calling command
G65 P9081 L Iterations R R point Z Z point;
A custom macro body is created as follows.

```
O9081 ;
#3003 = 1 ;
G00 Z#18 ;
G01 Z#26 ;
G00 Z-[ ROUND[#18] + ROUND[#26] ] ;
#3003 = 0 :
M99 ;
```

Disable single block stop
#18 corresponds to R and #26 to Z.

> **NOTE**
> #3003 is cleared by a reset.

- Enabling of feed hold, feedrate override, and exact stop check #3004 (Attribute: R/W)

Assigning the following values in system variable #3004 allows the specification of whether feed hold and feedrate override are enabled in the following blocks or whether exact stop in G61 mode or by G09 command is disabled.

Variable number and variable name	Value	Field hold	Feedrate override	Exact stop
#3004 [#_CNTL2]	0	Enabled	Enabled	Enabled
	1	Disabled	Enabled	Enabled
	2	Enabled	Disabled	Enabled
	3	Disabled	Disabled	Enabled
	4	Enabled	Enabled	Disabled
	5	Disabled	Enabled	Disabled
	6	Enabled	Disabled	Disabled
	7	Disabled	Disabled	Disabled

In addition, the following variable names can be used to enable or disable feed hold, feedrate override, and exact stop in G61 mode or by the G09 command, individually.

Variable number and variable name	Value	Feed hold	Feedrate override	Exact stop
[#_M_FHD]	0	Enabled	-	-
	1	Disabled	-	-
[#_M_OV]	0	-	Enabled	-
	1	-	Disabled	-
[#_M_EST]	0	-	-	Enabled
	1	-	-	Disabled

NOTE

1 These system variables are provided to maintain compatibility with conventional NC programs. It is recommended that functions provided by G63, G09, G61, and other G codes be used to enable or disable feed hold, feedrate override, and exact stop.

2 When the feed hold button is pressed during execution of a block for which feed hold is disabled:

<1> If the feed hold button is kept pressed, operation stops after execution of the block. When single block stop is disabled, however, operation does not stop.

<2> If the pressed feed hold button is released, the feed hold lamp lights, but operation does not stop until the end of the first block that was enabled.

3 #3004 is cleared by a reset.

4 If exact stop is disabled by #3004, the original exact stop position between cutting feed and positioning block is not affected. #3004 can temporarily disabled exact stop in G61 mode or by the G09 command between cutting feed and cutting feed.

- Settings #3005 (Attribute: R/W)

Settings can be read and written.
Binary values are converted to decimals.

#3005								
	#15	#14	#13	#12	#11	#10	#9	#8
Setting							FCV	
	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC

#9 (FCV) : Whether to use the FANUC Series 15 program format conversion capability
#5 (SEQ) : Whether to automatically insert sequence numbers
#2 (INI) : Millimeter input or inch input
#1 (ISO) : Whether to use EIA or ISO as the output code
#0 (TVC) : Whether to make a TV check

- Stop with a message #3006 (Attribute: W)

When "#3006=1 (MESSAGE);" is commanded in the macro, the program executes blocks up to the immediately previous one and then stops. When a message of up to 26 characters, which is enclosed by a control-in character and control-out character, is programmed in the same block, the message is displayed on the external operator message screen.

Variable number	Variable name	Description
#3006	[#_MSGSTP]	Stop with a message

- Status of a mirror image #3007 (Attribute: R)

The status of an mirror image (setting or DI) at that point in time can be obtained for each axis by reading #3007.

Variable number	Variable name	Description
#3007	[#_MRIMG]	Status of a mirror image

When the status is indicated in binary, each bit corresponds with an axis as follows.

Bit	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16
nth axis	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis	16	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1

For the 32 bits, 0 indicates that a mirror image is disabled and 1 indicates that a mirror image is enabled.

[Example] When #3007 is 3, a mirror image is enabled for the 1st and 2nd axes.

NOTE
1. The status of a programmable mirror image is not reflected on this variable.
2. When the mirror image function is set for the same axis by the mirror image signal and setting, the signal value and setting value are ORed and then output.
3. When mirror image signals for axes other than the controlled axes are turned on, they are not read into system variable #3007.

- Status during restart of a program #3008 (Attribute: R)

Whether a program is restarting can be determined by reading #3008.

Variable number	Variable name	Description
#3008	[#_PRSTR]	0: Program is not restarting. 1: Program is restarting.

- Time #3011, #3012 (Attribute: R)

Year/month/date and hour/minute/second can be obtained by reading system variables #3011 and #3012. This variable is read-only. To change year/month/date and hour/minute/second, use the timer screen.

[Example] May 20, 2004, PM 04:17:05
#3011 = 20040520
#3012 = 161705

- Total number of parts and the number of required parts #3901 and #3902 (Attribute: R/W)

The number of required parts and the number of machined parts can be displayed on the screen by using the operation time and part number displaying function. When the (total) number of machined parts reaches the number of required parts, a signal indicating the fact is sent to the machine (PMC side).
The system variables can be used to read or write the total number of parts and the number of required parts.

Variable number	Variable name	Description
#3901	[#_PRTSA]	Total number of parts
#3902	[#_PRTSN]	Number of required parts

- Type of tool compensation memory #3980 (Attribute: R)

M

System variable #3980 can be used to read the type of compensation memory.

Variable number	Variable name	Description
#3980	[#_OFSMEM]	Types of tool compensation memory 0: Tool compensation memory A 1: Tool compensation memory B 2: Tool compensation memory C

- Main program number #4000 (Attribute: R)

System variable #4000 can be used to read the main program number regardless of the level of a subprogram.

Variable number	Variable name	Description
#4000	[#_MAINO]	Main program number

NOTE

1. The main program number indicates the number of the program that is first started.
2. When an O number is specified by MDI during execution of the main program or when the second O number is specified in DNC mode, the value of #4000 changes to the specified O number. In addition, when no programs are registered or when no O numbers are specified in DNC mode, the value of #4000 changes to 0.

- Modal information #4001-#4130, #4201-#4330, #4401-#4530 (Attribute: R)

The modal information specified before the previous block of the macro statement that reads system variables #4001 to #4130 can be obtained in the block currently being looked ahead, by reading system variables #4001 to #4130.
The modal information of the block currently being executed can be obtained by reading system variables #4201 to #4330.
The modal information specified before the block interrupted by an interruption type custom macro by reading system variables #4401 to #4530.
The unit used when it was specified is applied.

M

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4001 #4201 #4401	[#_BUFG[1]] [#_ACTG[1]] [#_INTG[1]]	Modal information (G code: group 1)
<1> <2> <3>	#4002 #4202 #4402	[#_BUFG[2]] [#_ACTG[2]] [#_INTG[2]]	Modal information (G code: group 2)
: :	: :	: :	: :
<1> <2> <3>	#4030 #4230 #4430	[#_BUFG[30]] [#_ACTG[30]] [#_INTG[30]]	Modal information (G code: group 30)
<1> <2> <3>	#4102 #4302 #4502	[#_BUFB] [#_ACTB] [#_INTB]	Modal information (B code)
<1> <2> <3>	#4107 #4307 #4507	[#_BUFD] [#_ACTD] [#_INTD]	Modal information (D code)
<1> <2> <3>	#4108 #4308 #4508	[#_BUFE] [#_ACTE] [#_INTE]	Modal information (E code)
<1> <2> <3>	#4109 #4309 #4509	[#_BUFF] [#_ACTF] [#_INTF]	Modal information (F code)
<1> <2> <3>	#4111 #4311 #4511	[#_BUFH] [#_ACTH] [#_INTH]	Modal information (H code)
<1> <2> <3>	#4113 #4313 #4513	[#_BUFM] [#_ACTM] [#_INTM]	Modal information (M code)
<1> <2> <3>	#4114 #4314 #4514	[#_BUFN] [#_ACTN] [#_INTN]	Modal information (sequence number N)
<1> <2> <3>	#4115 #4315 #4515	[#_BUFO] [#_ACTO] [#_INTO]	Modal information (program number O)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4119 #4319 #4519	[#_BUFS] [#_ACTS] [#_INTS]	Modal information (S code)
<1> <2> <3>	#4120 #4320 #4520	[#_BUFT] [#_ACTT] [#_INTT]	Modal information (T code)
<1> <2> <3>	#4130 #4330 #4530	[#_BUFWZP] [#_ACTWZP] [#_INTWZP]	Modal information (additional workpiece coordinate system number P)

T

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1> <2> <3>	#4001 #4201 #4401	[#_BUFG[1]] [#_ACTG[1]] [#_INTG[1]]	Modal information (G code: group 1)
<1> <2> <3>	#4002 #4202 #4402	[#_BUFG[2]] [#_ACTG[2]] [#_INTG[2]]	Modal information (G code: group 2)
: :	: :	: :	: :
<1> <2> <3>	#4030 #4230 #4430	[#_BUFG[30]] [#_ACTG[30]] [#_INTG[30]]	Modal information (G code: group 30)
<1> <2> <3>	#4108 #4308 #4508	[#_BUFE] [#_ACTE] [#_INTE]	Modal information (E code)
<1> <2> <3>	#4109 #4309 #4509	[#_BUFF] [#_ACTF] [#_INTF]	Modal information (F code)
<1> <2> <3>	#4113 #4313 #4513	[#_BUFM] [#_ACTM] [#_INTM]	Modal information (M code)
<1> <2> <3>	#4114 #4314 #4514	[#_BUFN] [#_ACTN] [#_INTN]	Modal information (sequence number N)
<1> <2> <3>	#4115 #4315 #4515	[#_BUFO] [#_ACTO] [#_INTO]	Modal information (program number O)
<1> <2> <3>	#4119 #4319 #4519	[#_BUFS] [#_ACTS] [#_INTS]	Modal information (S code)
<1> <2> <3>	#4120 #4320 #4520	[#_BUFT] [#_ACTT] [#_INTT]	Modal information (T code)
<1> <2> <3>	#4130 #4330 #4530	[#_BUFWZP] [#_ACTWZP] [#_INTWZP]	Modal information (additional workpiece coordinate system number P)

NOTE

1. Previous block and running block
   Since the CNC reads the block that is ahead of the block currently being executed by the machining program, the block being retrieved by the CNC is normally different from that currently being executed. The previous block indicates the block that is ahead of the block being retrieved by the CNC, that is, the block that is ahead of the program block in which #4001 to #4130 are specified.
2. When variables exceeding the number of control axes are specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs.

[Example]

```
O1234 ;
N10 G00 X200. Y200. ;
N20 G01 X1000. Y1000. F10. ;
    :
    :
N50 G00 X500. Y500. ;
N60 #1 = #4001 ;
```

Assume that the CNC is currently executing N20. If the CNC retrieved and processed the blocks up to N60 as shown above, the running block is N20 and the previous block is N50. Therefore, group 1 modal information in the running block is G01 and group 1 modal information in the previous block is G00.

When N60 #1 = #4201, #1 = 1.

When N60 #1 = #4001, #1 = 0.

- Position information #5001-#5080, #100001-#100200 (Attribute: R)

The end position of the previous block, the specified current position (for the machine coordinate system and workpiece coordinate system), and the skip signal position can be obtained by reading the values of system variables #5001 to #5080 or #100001 to -#100200.

Variable number	Variable name	Position information	Coordinate system	Tool position/tool length/cutter compensation	Reading operation during movement
#5001 #5002 : #5020	[#_ABSIO[1]] [#_ABSIO[2]] : [#_ABSIO[20]]	1st axis block end point position 2nd axis block end point position : 20th axis block end point position	Workpiece coordinate system	Not included	Enabled
#100001 #100002 : #100050	[#_ABSIO[1]] [#_ABSIO[2]] : [#_ABSIO[50]]	1st axis block end point position 2nd axis block end point position : 20th axis block end point position			
#5021 #5022 : #5040	[#_ABSMT[1]] [#_ABSMT[2]] : [#_ABSMT[20]]	1st axis current position 2nd axis current position : 20th axis current position	Machine coordinate system	Included	Disabled
#100051 #100052 : #100100	[#_ABSMT[1]] [#_ABSMT[2]] : [#_ABSMT[50]]	1st axis current position 2nd axis current position : 50th axis current position			
#5041 #5042 : #5060	[#_ABSOT[1]] [#_ABSOT[2]] : [#_ABSOT[20]]	1st axis current position 2nd axis current position : 20th axis current position	Workpiece coordinate system	Included	Disabled
#100101 #100102 : #100150	[#_ABSOT[1]] [#_ABSOT[2]] : [#_ABSOT[50]]	1st axis current position 2nd axis current position : 50th axis current position			
#5061 #5062 : #5080	[#_ABSKP[1]] [#_ABSKP[2]] : [#_ABSKP[20]]	1st axis skip position 2nd axis skip position : 20th axis skip position	Workpiece coordinate system	Included	Enabled
#100151 #100152 : #100200	[#_ABSKP[1]] [#_ABSKP[2]] : [#_ABSKP[50]]	1st axis skip position 2nd axis skip position : 50th axis skip position			

NOTE

1. When variables exceeding the number of control axes are specified, the alarm (PS0115) “VARIABLE NO. OUT OF RANGE” occurs.
2. The position information for 20th or earlier axis can be used with #5001 to #5080.
3. The block end point position (ABSIO) of the skip (G31) is the position where the skip signal is turned on. If the skip signal is not turned on, the position is the end position of the block.
4. "Read operation during movement is disabled" means that the accurate reading of values during movement is not guaranteed.

- Tool length compensation value #5081-#5100, #100201-#100250 (Attribute: R)

M

Tool length compensation in the block currently being executed can be obtained for each axis by reading system variables #5081 to #5100 or #100201 to #100250.

Variable number	Variable name	Position information	Read operation during movement
#5081	[#_TOFS[1]]	1st axis tool length compensation value	Disabled
#5082	[#_TOFS[2]]	2nd axis tool length compensation value	
:	:	:	
#5100	[#_TOFS[20]]	20th axis tool length compensation value	
#100201	[#_TOFS[1]]	1st axis tool length compensation value	
#100202	[#_TOFS[2]]	2nd axis tool length compensation value	
:	:	:	
#100250	[#_TOFS[50]]	50th axis tool length compensation value	

NOTE

1. When variables exceeding the number of control axes are specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs.
2. The tool length compensation for 20th or earlier axis can be used with #5081 to #5100.

- Tool offset #5081-#5083, #5121-#5123 (Attribute: R)

T

Tool offset in the block currently being executed can be obtained for each axis by reading system variables #5081 to #5083 or #5121 to #5123. (X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes)

<1> Without tool geometry/wear compensation memory

Variable number	Variable name	Position information	Read operation during movement
#5081 #5082 #5083	[#_TOFSWX] [#_TOFSWZ] [#_TOFSWY]	X-axis tool offset value Z-axis tool offset value Y-axis tool offset value	Disabled

<2> With tool geometry/wear compensation memory

Variable number	Variable name	Position information	Read operation during movement
#5081 #5082 #5083	[#_TOFSWX] [#_TOFSWZ] [#_TOFSWY]	X-axis tool offset value Z-axis tool offset value Y-axis tool offset value	Disabled
#5121 #5122 #5123	[#_TOFSGX] [#_TOFSGZ] [#_TOFSGY]	X-axis tool offset value (geometry) Z-axis tool offset value (geometry) Y-axis tool offset value (geometry)	

When tool geometry/wear compensation memory exists, the system variable values vary as follows depending on parameter LWT (No.5002#2) and parameter LGT (No.5002#4).

Variable number	LWT=0 LGT=0	LWT=1 LGT=0	LWT=0 LGT=1	LWT=1 LGT=1
#5081 #5082 #5083	Wear compensation	0	Wear compensation	Wear compensation
#5121 #5122 #5123	Geometry compensation	Wear compensation + Geometry compensation	Geometry compensation	Geometry compensation

NOTE

1 The set value is read as the tool offset regardless of parameters ORC (No.5004#1) and OWD (No.5040#0).

2 To read the tool offset (geometry) using #5121 to #5123, set parameter VHD (No.6004#2) to 0.

- Servo position deviation #5101-#5120, #100251-#100300 (Attribute: R)

The servo position deviation for each axis can be obtained by reading system variables #5101 to #5120 or #100251 to #100300.

Variable number	Variable name	Position information	Read operation during movement
#5101	[#_SVERR[1]]	1st axis servo position deviation	Disabled
#5102	[#_SVERR[2]]	2nd axis servo position deviation	
:	:	:	
#5120	[#_SVERR[20]]	20th axis servo position deviation	
#100251	[#_SVERR[1]]	1st axis servo position deviation	
#100252	[#_SVERR[2]]	2nd axis servo position deviation	
:	:	:	
#100300	[#_SVERR[50]]	50th axis servo position deviation	

NOTE

1. When variables exceeding the number of control axes are specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs.
2. The servo position deviation for 20th or earlier axis can be used with #5101 to #5120.

- Manual handle interruption #5121-#5140, #100651-#100700 (Attribute: R)

The manual handle interruption for each axis can be obtained by reading system variables #5121 to #5140 or #100651 to #100700.

Variable number	Variable name	Position information	Read operation during movement
#5121	[#_MIRTP[1]]	1st axis manual handle interruption	Disabled
#5122	[#_MIRTP[2]]	2nd axis manual handle interruption	
:	:	:	
#5140	[#_MIRTP[20]]	20th axis manual handle interruption	
#100651	[#_MIRTP[1]]	1st axis manual handle interruption	
#100652	[#_MIRTP[2]]	2nd axis manual handle interruption	
:	:	:	
#100700	[#_MIRTP[50]]	50th axis manual handle interruption	

NOTE

1. When variables exceeding the number of control axes are specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs.
2. The manual handle interruption for 20th or earlier axis can be used with #5121 to #5140.

T

NOTE

#5121 to #5140 are enabled only when parameter VHD (No.6004#2) is set to 1.

- Distance to go #5181-#5200, #100801-#100850 (Attribute: R)

The distance to go value for each axis can be obtained by reading system variables #5181 to #5200 or #100801 to #100850.

Variable number	Variable name	Position information	Read operation during movement
#5181	[#_DIST[1]]	1st axis distance to go value	Disabled
#5182	[#_DIST[2]]	2nd axis distance to go value	
:	:	:	
#5200	[#_DIST[20]]	20th axis distance to go value	
#100801	[#_DIST[1]]	1st axis distance to go value	
#100802	[#_DIST[2]]	2nd axis distance to go value	
:	:	:	
#100850	[#_DIST[50]]	50th axis distance to go value	

NOTE

1. When variables exceeding the number of control axes are specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs.
2. The distance to go values for 20th or earlier axis can be used with #5181 to #5200.

- Workpiece coordinate system shift value #2501, #2601 (Attribute: R/W)

T

The X-axis workpiece coordinate system shift value can be obtained by system variable #2501 while Z-axis workpiece coordinate system shift value by system variable #2601. The X-axis or Z-axis workpiece coordinate system shift value can be changed by assigning values to the system variables. (X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes)

Variable number	Variable name	Description
#2501	[#_WZ_SFTX]	X-axis workpiece shift value
#2601	[#_WZ_SHTZ]	Z-axis workpiece shift value

- Workpiece origin offset value #5201-#5340, #100301-#100650 (Attribute: R/W)

The workpiece origin offset value can be obtained by reading system variables #5201 to #5340 or #100301 to #100650. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#5201 #5202 : #5220	[#_WZCMN[1]] [#_WZCMN[2]] : [#_WZCMN[20]]	1st axis common workpiece origin offset value 2nd axis common workpiece origin offset value : 20th axis common workpiece origin offset value (*1)	Common workpiece origin offset value (applied to all coordinate systems) (*1)
#5221 #5222 : #5240	[#_WZG54[1]] [#_WZG54[2]] : [#_WZG54[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G54
#5241 #5242 : #5260	[#_WZG55[1]] [#_WZG55[2]] : [#_WZG55[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G55
#5261 #5262 : #5280	[#_WZG56[1]] [#_WZG56[2]] : [#_WZG56[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G56
#5281 #5282 : #5300	[#_WZG57[1]] [#_WZG57[2]] : [#_WZG57[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G57
#5301 #5302 : #5320	[#_WZG58[1]] [#_WZG58[2]] : [#_WZG58[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G58
#5321 #5322 : #5340	[#_WZG59[1]] [#_WZG59[2]] : [#_WZG59[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	G59
#100301 #100302 : #100350	[#_WZCMN[1]] [#_WZCMN[2]] : [#_WZCMN[50]]	1st axis common workpiece origin offset value 2nd axis common workpiece origin offset value : 50th axis common workpiece origin offset value	Common workpiece origin offset value (applied to all coordinate systems)
#100351 #100352 : #100400	[#_WZG54[1]] [#_WZG54[2]] : [#_WZG54[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	G54
#100401 #100402 : #100450	[#_WZG55[1]] [#_WZG55[2]] : [#_WZG55[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	G55
#100451 #100452 : #100500	[#_WZG56[1]] [#_WZG56[2]] : [#_WZG56[50]]	1st axis workpiece origin offset 2nd axis workpiece origin offset : 50th axis workpiece origin offset	G56

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#100501 #100502 : #100550	[#_WZG57[1]] [#_WZG57[2]] : [#_WZG57[50]]	1st axis workpiece origin offset 2nd axis workpiece origin offset : 50th axis workpiece origin offset	G57
#100551 #100552 : #100600	[#_WZG58[1]] [#_WZG58[2]] : [#_WZG58[50]]	1st axis workpiece origin offset 2nd axis workpiece origin offset : 50th axis workpiece origin offset	G58
#100601 #100602 : #100650	[#_WZG59[1]] [#_WZG59[2]] : [#_WZG59[50]]	1st axis workpiece origin offset 2nd axis workpiece origin offset : 50th axis workpiece origin offset	G59

(*1) When the path control type is the lathe system type, the external workpiece origin offset is assumed.

M

The following variables can also be used when bit 5 (D15) of parameter No. 6004 is set to 0:

Axis	Function	Variable number
1st axis	Common workpiece origin offset value	#2500
	G54 workpiece origin offset value	#2501
	G55 workpiece origin offset value	#2502
	G56 workpiece origin offset value	#2503
	G57 workpiece origin offset value	#2504
	G58 workpiece origin offset value	#2505
	G59 workpiece origin offset value	#2506
2nd axis	Common workpiece origin offset value	#2600
	G54 workpiece origin offset value	#2601
	G55 workpiece origin offset value	#2602
	G56 workpiece origin offset value	#2603
	G57 workpiece origin offset value	#2604
	G58 workpiece origin offset value	#2605
	G59 workpiece origin offset value	#2606
3rd axis	Common workpiece origin offset value	#2700
	G54 workpiece origin offset value	#2701
	G55 workpiece origin offset value	#2702
	G56 workpiece origin offset value	#2703
	G57 workpiece origin offset value	#2704
	G58 workpiece origin offset value	#2705
	G59 workpiece origin offset value	#2706
4th axis	Common workpiece origin offset value	#2800
	G54 workpiece origin offset value	#2801
	G55 workpiece origin offset value	#2802
	G56 workpiece origin offset value	#2803
	G57 workpiece origin offset value	#2804
	G58 workpiece origin offset value	#2805
	G59 workpiece origin offset value	#2806

T

The following variables can be used to maintain compatibility with conventional models.

Axis	Function	Variable number
1st axis	External workpiece origin offset value G54 workpiece origin offset value G55 workpiece origin offset value G56 workpiece origin offset value G57 workpiece origin offset value G58 workpiece origin offset value G59 workpiece origin offset value	#2550 #2551 #2552 #2553 #2554 #2555 #2556
2nd axis	External workpiece origin offset value G54 workpiece origin offset value G55 workpiece origin offset value G56 workpiece origin offset value G57 workpiece origin offset value G58 workpiece origin offset value G59 workpiece origin offset value	#2650 #2651 #2652 #2653 #2654 #2655 #2656
3rd axis	External workpiece origin offset value G54 workpiece origin offset value G55 workpiece origin offset value G56 workpiece origin offset value G57 workpiece origin offset value G58 workpiece origin offset value G59 workpiece origin offset value	#2750 #2751 #2752 #2753 #2754 #2755 #2756
4th axis	External workpiece origin offset value G54 workpiece origin offset value G55 workpiece origin offset value G56 workpiece origin offset value G57 workpiece origin offset value G58 workpiece origin offset value G59 workpiece origin offset value	#2850 #2851 #2852 #2853 #2854 #2855 #2856

NOTE

1 When variables exceeding the number of control axes are specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs.

2 The workpiece origin offset value for 20th or earlier axis can be used with #5201 to #5340.

M

NOTE

To use variables #2500 to #2806, #5201 to #5328, and #100301 to #100650, optional variables for the workpiece coordinate systems are necessary.

T

> **NOTE**
> To use variables #2550 to #2856, #5201 to #5340, and #100301 to #100650, optional variables for the workpiece coordinate systems are necessary.

- Workpiece origin offset value of the additional workpiece coordinate system

#7001-#7960, #101001-#116000 (Attribute: R/W)

M

#14001-#20000 (Attribute: R/W)

The workpiece origin offset value of the additional workpiece coordinate system can be obtained by reading system variables #7001 to #7960, #14001 to #20000, #101001 to #116000. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Additional workpiece system number
#7001 #7002 : #7020	[#_WZP1[1]] [#_WZP1[2]] : [#_WZP1[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	1 (G54.1 P1)
#7021 #7022 : #7040	[#_WZP2[1]] [#_WZP2[2]] : [#_WZP2[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	2 (G54.1 P2)
#7041 #7042 : #7060	[#_WZP3[1]] [#_WZP3[2]] : [#_WZP3[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	3 (G54.1 P3)
:	:	:	:
#7941 #7942 : #7960	[#_WZP48[1]] [#_WZP48[2]] : [#_WZP48[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	48 (G54.1 P48)

System variable number = 7000 + (Coordinate system number -1) × 20 + Axis number
Coordinate number: 1 to 48
Axis number: 1 to 20

M

Variable number	Variable name	Controlled axis	Additional workpiece system number
#14001 #14002 : #14020	[#_WZP1[1]] [#_WZP1[2]] : [#_WZP1[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	1 (G54.1 P1)
#14021 #14022 : #14040	[#_WZP2[1]] [#_WZP2[2]] : [#_WZP2[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	2 (G54.1 P2)
#14041 #14042 : #14060	[#_WZP3[1]] [#_WZP3[2]] : [#_WZP3[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	3 (G54.1 P3)
:	:	:	:
#19971 #19972 : #20000	[#_WZP300[1]] [#_WZP300[2]] : [#_WZP300[20]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 20th axis workpiece origin offset value	300 (G54.1 P300)

System variable number = 14000 + (Coordinate system number -1) × 20 + Axis number
Coordinate number: 1 to 300
Axis number: 1 to 20

Variable number	Variable name	Controlled axis	Additional workpiece system number
#101001 #101002 : #101050	[#_WZP1[1]] [#_WZP1[2]] : [#_WZP1[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	1 (G54.1 P1)
#101051 #101052 : #101100	[#_WZP2[1]] [#_WZP2[2]] : [#_WZP2[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	2 (G54.1 P2)
#101101 #101102 : #101150	[#_WZP3[1]] [#_WZP3[2]] : [#_WZP3[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	3 (G54.1 P3)
:	:	:	:
#115951 #115952 : #116000	[#_WZP300[1]] [#_WZP300[2]] : [#_WZP300[50]]	1st axis workpiece origin offset value 2nd axis workpiece origin offset value : 50th axis workpiece origin offset value	300 (G54.1 P300)

System variable number = 101000 + (Coordinate system number -1) × 50 + Axis number
Coordinate number: 1 to 300
Axis number: 1 to 50

M

> **NOTE**
> 1 When variables exceeding the number of control axes are specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs.
> 2 The workpiece origin offset of additional workpiece coordinate system for 20th or earlier axis can be used with #7001 to #7960 or #14001 to #20000.
> 3 Optional variables for 48 additional workpiece coordinate systems are #7001 to #7960 (G54.1 P1 to G54.1 P48). Optional variables for 300 additional workpiece coordinate systems are #14001 to #20000, and #101001 to #116000 (G54.1 P1 to G54.1 P300). With these variables, #7001 to #7960 can also be used.

T

> **NOTE**
> 1 When variables exceeding the number of control axes are specified, the alarm (PS0115) "VARIABLE NO. OUT OF RANGE" occurs.
> 2 Optional variables for 48 additional workpiece coordinate systems are #7001 to #7960 (G54.1 P1 to G54.1 P48).
> 3 Optional variables for 300 additional workpiece coordinate systems are #101001 to #116000 (G54.1 P1 to G54.1 P300). With these variables, #7001 to #7960 can also be used.

- Skip position (detection unit) #5421-#5440, #100701-#100750 (Attribute: R)

The skip position with the detection unit can be obtained by reading system variables #5421 to #5440 or #100701 to #100750. These values are integers.

Variable number	Variable name	Position information	Read operation during movement
#5421 #5422 : #5440	[#_SKPDTC[1]] [#_SKPDTC[2]] : [#_SKPDTC[20]]	1st axis skip position (detection unit) 2nd axis skip position (detection unit) : 20th axis skip position (detection unit)	Disabled
#100151 #100152 : #100200	[#_SKPDTC[1]] [#_SKPDTC[2]] : [#_SKPDTC[50]]	1st axis skip position (detection unit) 2nd axis skip position (detection unit) : 50th axis skip position (detection unit)	

NOTE
1. When variables exceeding the number of control axes are specified, the alarm (PS0115) “VARIABLE NO. OUT OF RANGE” occurs.
2. The skip position (detection unit) for 20th or earlier axis can be used with #5421 to #5440.

- Reference fixture offset number being selected #5500 (Attribute: R)

M

The reference fixture offset number being selected can be read by reading system variables #5500.

Variable number	Variable name	Description
#5500	[#_FOFSP]	Reference fixture offset number being selected

- Reference fixture offset value being selected #5501-#5520, #117001-#117050 (Attribute: R)

M

The reference fixture offset being selected can be read by reading system variables #5501 to #5520 or #117001# to 117050.

Variable number	Variable name	Description
#5501	[#_FOFSVAL[1]]	1st axis reference fixture offset value being selected
#5502	[#_FOFSVAL[2]]	2nd axis reference fixture offset value being selected
:	:	:
#5520	[#_FOFSVAL[20]]	20th axis reference fixture offset value being selected
#117001	[#_FOFSVAL[1]]	1st axis reference fixture offset value being selected
#117002	[#_FOFSVAL[2]]	2nd axis reference fixture offset value being selected
:	:	:
#117050	[#_FOFSVAL[50]]	50th axis reference fixture offset value being selected

NOTE
1. When variables exceeding the number of control axes are specified, the alarm (PS0115) “VARIABLE NO. OUT OF RANGE” occurs.
2. The reference fixture offset for 20th or earlier axis can be used with #5501 to #5520.

- Reference fixture offset value #5521-#5680, #117051-#117450 (Attribute: R/W)

M

The reference fixture offset values in the rotary table dynamic fixture offset function by reading system variables #5521 to #5680 or #117051 to #117450. The reference fixture offset values can also be changed by assigning values to the system values.

Variable number	Variable name	Controlled axis	Fixture offset number
#5521 #5522 : #5540	[#_FOFS1[1]] [#_FOFS1[2]] : [#_FOFS1[20]]	1st axis reference fixture offset value being selected 2nd axis reference fixture offset value being selected : 20th axis reference fixture offset value being selected	1 (G54.2 P1)
#5541 #5542 : #5560	[#_FOFS2[1]] [#_FOFS2[2]] : [#_FOFS2[20]]	1st axis reference fixture offset value being selected 2nd axis reference fixture offset value being selected : 20th axis reference fixture offset value being selected	2 (G54.2 P2)
:	:	:	:
#5661 #5662 : #5680	[#_FOFS8[1]] [#_FOFS8[2]] : [#_FOFS8[20]]	1st axis reference fixture offset value being selected 2nd axis reference fixture offset value being selected : 20th axis reference fixture offset value being selected	8 (G54.2 P8)
#117051 #117052 : #117100	[#_FOFS1[1]] [#_FOFS1[2]] : [#_FOFS1[50]]	1st axis reference fixture offset value being selected 2nd axis reference fixture offset value being selected : 50th axis reference fixture offset value being selected	1 (G54.2 P1)
#117101 #117102 : #117150	[#_FOFS2[1]] [#_FOFS2[2]] : [#_FOFS2[50]]	1st axis reference fixture offset value being selected 2nd axis reference fixture offset value being selected : 50th axis reference fixture offset value being selected	2 (G54.2 P2)
:	:	:	:
#117401 #117402 : #117450	[#_FOFS8[1]] [#_FOFS8[2]] : [#_FOFS8[50]]	1st axis reference fixture offset value being selected 2nd axis reference fixture offset value being selected : 50th axis reference fixture offset value being selected	8 (G54.2 P8)

NOTE

1 When variables exceeding the number of control axes are specified, the alarm (PS0115) “VARIABLE NO. OUT OF RANGE” occurs.

2 The reference fixture offset values for 20th or earlier axis can be used with #5521 to #5680.

- Dynamic reference tool compensation value #118051-#118450 (Attribute: R/W)

M

The dynamic reference tool compensation value in the rotary head dynamic tool compensation function can be obtained by reading system variables #118051 to #118450. The dynamic reference tool compensation value can also be obtained by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Dynamic tool offset number
#118051 #118052 : #118100	[#_DOFS1[1]] [#_DOFS1[2]] : [#_DOFS1[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	1 (G43.2H1)
#118101 #118102 : #118150	[#_DOFS2[1]] [#_DOFS2[2]] : [#_DOFS2[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	2 (G43.2H2)
#118151 #118152 : #118200	[#_DOFS3[1]] [#_DOFS3[2]] : [#_DOFS3[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	3 (G43.2H3)
#118201 #118202 : #118250	[#_DOFS4[1]] [#_DOFS4[2]] : [#_DOFS4[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	4 (G43.2H4)
#118251 #118252 : #118300	[#_DOFS5[1]] [#_DOFS5[2]] : [#_DOFS5[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	5 (G43.2H5)
:	:	:	:
#118401 #118402 : #118450	[#_DOFS8[1]] [#_DOFS8[2]] : [#_DOFS8[50]]	1st axis dynamic reference tool compensation value 2nd axis dynamic reference tool compensation value : 50th axis dynamic reference tool compensation value	8 (G43.2H8)

NOTE

When variables exceeding the number of control axes are specified, the alarm (PS0115) “VARIABLE NO. OUT OF RANGE” occurs.

- Switching between P-CODE variables and system variables (#10000-) #8570 (Attribute: R/W)

This system variable allows read/write operations of P-CODE variables (#10000 to #89999) for the macro executor function. For details on P-CODE variables, refer to the Macro Compiler / Macro Executor Programming Guide (B-63943E-2).
System variable #8570 can be used to make variables #10000 to #89999 correspond to either P-CODE variables or system variables.

#8570 setting	Specified variable	Corresponding variable
#8570 = 0	#10000 : #89999	P-CODE variables (#10000) : P-CODE variables (#89999)
#8570 = 1	#10000 : #89999	System variables (#10000) : System variables (#89999)

Example

#8570 = 0 ;
#10000 = 123 ; → Writing to system variable #10000 (tool compensation)
#8570 = 1 ;
#10000 = 456 ; → Writing to P-CODE variable #10000 (tool compensation)

NOTE

1 Variable #8570 can be used only when the macro executor function is enabled.
2 System variables (#10000-) always correspond to system variables specified by their variable names even when #8570 is 1.
3 When an attempt is made to access a variable that cannot be used with P-CODE variables (#10000-), an alarm (PS0115) occurs.

16.3 ARITHMETIC AND LOGIC OPERATION

Various operations can be performed on variables. Program an arithmetic and logic operation in the same way as for a general arithmetic expression.

#i=<expression>

<Expression>

The expression to the right of the arithmetic and logic operation contains constants and/or variables combined by a function or operator. Variables #j and #k below can be replaced with a constant. If a constant used in an expression has no decimal point, it is assumed to end with a decimal point.

Table 16.3 (a) Arithmetic and logic operation

Type of operation	Operation	Description
<1> Definition or replacement	#i=#j	Definition or replacement of a variable
<2> Addition-type operations	#i=#j+#k #i=#j-#k #i=#j OR #k #i=#j XOR #k	Addition Subtraction Logical OR (bit by bit of 32 bits) Exclusive OR (bit by bit of 32 bits)
<3> Multiplication-type operations	#i=#j*#k #i=#j/#k #i=#j AND #k #i=#j MOD #k	Multiplication Division Logical AND (bit by bit of 32 bits) Remainder (A remainder is obtained after #j and #k are rounded to their nearest whole numbers. When #j is a negative value, #i is assumed to be a negative value.)
<4> Functions	#i=SIN[#j] #i=COS[#j] #i=TAN[#j] #i=ASIN[#j] #i=ACOS[#j] #i=ATAN[#j] #i=ATAN[#j]/[#k] #i=ATAN[#j,#k] #i=SQRT[#j] #i=ABS[#j] #i=BIN[#j] #i=BCD[#j] #i=ROUND[#j] #i=FIX[#j] #i=FUP[#j] #i=LN[#j] #i=EXP[#j] #i=POW[#j,#k] #i=ADP[#j]	Sine (in degrees) Cosine (in degrees) Tangent (in degrees) Arc sine Arc cosine Arc tangent (one argument), ATN can also be used. Arc tangent (two arguments), ATN can also be used. Arc tangent (two arguments), ATN can also be used. Square root, SQR can also be used. Absolute value Conversion from BCD to binary Conversion from binary to BCD Rounding off, RND can also be used. Rounding down to an integer Rounding up to an integer Natural logarithm Exponent using base e (2.718...) Power (#j to the #kth power) Addition of a decimal point

Explanation

- Angle units

The units of angles used with the SIN, COS, ASIN, ACOS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

- ARCSIN #i = ASIN[#j];

- The solution ranges are as indicated below:
  When the NAT bit (bit 0 of parameter 6004) is set to 0: 270° to 90°
  When the NAT bit (bit 0 of parameter 6004) is set to 1: -90° to 90°
- When #j is beyond the range of -1 to 1, an alarm PS0119 is issued.
- A constant can be used instead of the #j variable.

- ARCCOS #i = ACOS[#j];

- The solution ranges from 180° to 0°.
- When #j is beyond the range of -1 to 1, an alarm PS0119 is issued.
- A constant can be used instead of the #j variable.

- ARCTAN #i = ATAN[#j]/[#k]; (two arguments)

- ATAN[#j,#k] is equivalent to ATAN[#j]/[#k].
- When point (#k,#j) on plane X-Y is given, this function returns the value of the arc tangent for the angle made by the point.
- A constant can be used instead of the #j variable.
- The solution ranges are as follows:
  When the NAT bit (bit 0 of parameter 6004) is set to 0: 0° to 360°

Example:

When #1 = ATAN[-1]/[-1]; is specified, #1 is 225.0.

When the NAT bit (bit 0 of parameter 6004) is set to 1: -180° to 180°

Example:

When #1 = ATAN[-1]/[-1]; is specified, #1 is -135.0.

- ARCTAN #i = ATAN[#j]; (one argument)

- When ATAN is specified with one argument, this function returns the main value of arc tangent (-90° ≤ ATAN[#j] ≤ 90°). In other word, this function returns the same value as ATAN in calculator specifications.
- To use this function as the dividend of a division, be sure to enclose it with brackets ([]). If this function is not enclosed, ATAN[#j]/[#k] is assumed.

Example:

#100 = [ATAN[1]]/10 ;	:	Divides ATAN with one argument by 10.
#100 = ATAN[1]/[10] ;	:	Executes ATAN with two arguments.
#100 = ATAN[1]/10 ;	:	Assumes ATAN with two arguments, but issues an alarm PS1131 because the X coordinate specification is not enclosed with brackets ([]).

- Natural logarithm #i = LN[#j];

- When the antilogarithm (#j) is zero or smaller, an alarm PS0119 is issued.
- A constant can be used instead of the #j variable.

- Exponential function #i = EXP[#j];

- When the result of the operation overflows, an alarm PS0119 is issued.
- A constant can be used instead of the #j variable.

- ROUND function

- When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.

Example:

When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.

- When the ROUND function is used in NC statement addresses, the ROUND function rounds off the specified value according to the least input increment of the address.

Example:

Creation of a drilling program that cuts according to the values of variables #1 and #2, then returns to the original position

Suppose that the increment system is 1/1000 mm, variable #1 holds 1.2345, and variable #2 holds 2.3456. Then,

G00 G91 X-#1;	Moves 1.235 mm in negative direction.
G01 X-#2 F300;	Moves 2.346 mm in negative direction.
G00 X[#1+#2];	Since 1.2345 + 2.3456 = 3.5801 in positive direction, the travel distance is 3.580, which does not return the tool to the original position.

This difference comes from whether addition is performed before or after rounding off. G00X-[ROUND[#1]+ROUND[#2]]; must be specified to return the tool to the original position.

- Add decimal point (ADP) function

- ADP[#n] (n = 1 to 33) can be executed to add a decimal point to an argument passed with no decimal point, in the subprogram.

Example:

In the subprogram called with G65 P_X10;, the value of ADP[#24] is a value to which a decimal point is added at its end (that is, 10.). Use this function when you do not want to consider the increment system in the subprogram. When bit 4 (CVA) of parameter No. 6007 is set to 1, however, the ADP function cannot be used because any argument is converted to 0.01 the moment it is passed.

> **NOTE**
> For compatibility among programs, it is recommended that the ADP function be not used, and decimal points be added in the argument specification for a macro call.

- Rounding up and down to an integer (FUP and FIX)

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handling negative numbers.

Example:

Suppose that #1=1.2 and #2=-1.2.
When #3=FUP[#1] is executed, 2.0 is assigned to #3.
When #3=FIX[#1] is executed, 1.0 is assigned to #3.
When #3=FUP[#2] is executed, -2.0 is assigned to #3.
When #3=FIX[#2] is executed, -1.0 is assigned to #3.

- Abbreviations of arithmetic and logic operation commands

When a function is specified in a program, the first two characters of the function name can be used to specify the function.

Example:

ROUND → RO
FIX → FI

> **NOTE**
> POW cannot be abbreviated.

- Priority of operations

<1> Functions
<2> Operations such as multiplication and division (*, /, AND)
<3> Operations such as addition and subtraction (+, -, OR, XOR)

Example) $\#1=\underbrace{\#2+\underbrace{\#3*\underbrace{SIN[\#4]}_{<1>}}_{<2>}}_{<3>};$

<1>, <2> and <3> indicate the order of operations.

- Bracket nesting

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, an alarm PS0118 occurs.

Example) $\#1=\underbrace{SIN\ \underbrace{[\ \underbrace{[\ \underbrace{[\underbrace{\#2+\#3}_{<1>}]\ *\#4}_{<2>}\ +\#5]}_{<3>}\ *\#6]}_{<4>}}_{<5>};$

<1> to <5> indicate the order of operations.

Limitation

- **Caution concerning decreased precision**
  When bit 0 (F16) of parameter No. 6008 is set to 0

- Addition and subtraction
  Note that when an absolute value is subtracted from another absolute value in addition or subtraction, the relative error may become 10^{-15} or greater.
  For example, assume that #1 and #2 have the following true values in the process of operation.
  (The following values are examples in the process of operation and cannot actually be specified from any program.)
```
#1=9876543210.987654321
#2=9876543210.987657777
```
  You cannot obtain the following result with operation #2-#1:
```
#2-#1=0.000003456
```
  This is because the precision of custom macro variables is 15 decimal digits. With this precision, the values of #1 and #2 become:
```
#1=9876543210.987650000
#2=9876543210.987660000
```
  (Precisely, the actual values are slightly different from the above values because they are internally processed in binary.) Therefore, the result is:
```
#2-1=0.000010000
```
  A large error occurs.
- Logical expressions
  Be aware of errors that can result from conditional expressions using EQ, NE, GT, LT, GE, and LE because they are processed basically in the same way as addition and subtraction. For example, if the following statement is used to decide whether #1 is equal to #2 in the above example, a correct decision may not be resulted because errors may occur:
```
IF [#1 EQ #2]
```
  Evaluate the difference between #1 and #2 with:
```
IF [ABS [#1-#2]LT 0.1]
```
  Then, assume that the values are equal when the difference does not exceed the allowable error range.
- Trigonometric functions
  The absolute error is guaranteed for trigonometric functions. However, the relative error is 10^{-15} or greater. Carefully perform multiplication or division after executing a trigonometric function.
- FIX function
  When using the FIX function for the result of an operation, be careful with the precision. For example, when the following operations are performed, the value of #3 may not always be 2.
```
N10 #1=0.002;
N20 #2=#1*1000;
N30 #3=FIX[#2];
```

This is because an error may occur in operation N20 and the result may not be

```
#2=2.0000000000000000
```

but a value a little smaller than 2 such as the following:

```
#2=1.9999999999999997
```

To prevent this, specify N30 as follows:

```
N30 #3=FIX[#2+0.001];
```

Generally, specify the FIX function as follows:

FIX[expression] → FIX[expression ±ε]

(Specify +ε when the value of the expression is positive or -ε when it is negative, and 0.1, 0.01, 0.001, ... for ε as required.)

> **NOTE**
> The operation result of exponential function #i=EXP[#j]; overflows when #j exceeds about 790.

When bit 0 (F16) of parameter No. 6008 is set to 1

Errors may occur when operations are performed.

Table 16.3 (b) Errors involved in operations

Operation	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error(*1) $\left\lvert \frac{\varepsilon}{a} \right\rvert$
a = b / c	4.66×10^{-10}	1.88×10^{-9}	
a = $\sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a = b + c a = b – c	2.33×10^{-10}	5.32×10^{-10}	(*2) MIN $\left\lvert \frac{\varepsilon}{b} \right\rvert$ $\left\lvert \frac{\varepsilon}{c} \right\rvert$
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error(*3) $\lvert \varepsilon \rvert$ degrees
a = ATAN [b] / [c]	1.8×10^{-6}	3.6×10^{-6}	

> **NOTE**
> 1 The relative error depends on the result of the operation.
> 2 Smaller of the two types of errors is used.
> 3 The absolute error is constant, regardless of the result of the operation.
> 4 Function TAN performs SIN/COS.
> 5 Note that, in the case of natural logarithm #i=LN[#j]; and exponential function #i=EXP[#j];, the relative error may become 10^{-8} or greater.
> 6 The operation result of exponential function #i=EXP[#j]; overflows when #j exceeds about 110.

- The precision of variable values is about 8 decimal digits. When very large numbers are handled in an addition or subtraction, the expected results may not be obtained.

Example:

When an attempt is made to assign the following values to variables #1 and #2:

#1=9876543210123.456

#2=9876543277777.777

the values of the variables become:

#1=9876543200000.000

#2=9876543300000.000

In this case, when #3=#2-#1; is calculated, #3=100000.000 results. (The actual result of this calculation is slightly different because it is performed in binary.)

- Also be aware of errors that can result from conditional expressions using EQ, NE, GE, GT, LE, and LT.

Example:

IF[#1 EQ #2] is effected by errors in both #1 and #2, possibly resulting in an incorrect decision.

Therefore, instead find the difference between the two variables with IF[ABS[#1-#2]LT0.001].

Then, assume that the values of the two variables are equal when the difference does not exceed an allowable limit (0.001 in this case).

- Also, be careful when rounding down a value.

Example:

When #2=#1*1000; is calculated where #1=0.002;, the resulting value of variable #2 is not exactly 2 but 1.99999997.

Here, when #3=FIX[#2]; is specified, the resulting value of variable #3 is not 2.0 but 1.0.

In this case, round down the value after correcting the error so that the result is greater than the expected number, or round it off as follows:

#3=FIX[#2+0.001]

#3=ROUND[#2]

- Brackets

Brackets ([,]) are used to enclose an expression.
Note that parentheses are used for comments.

- Divisor

When a divisor of zero is specified in a division, an alarm PS0112 occurs.

16.4 INDIRECT AXIS ADDRESS SPECIFICATION

Overview

When the custom macro function is enabled, you can use AX[(axis-number)] in an axis address specification to indirectly specify an axis with its axis number and not to directly specify it with its axis name.
You can also use AXNUM[(axis-name)] to obtain the axis number for an axis name.

Explanation

- Indirect axis address

You can use indirect axis address AX[ ] to specify an axis with its axis number. (An equal sign (=) is always required following AX[ ].)

AX[(axis-number)] = (numeric-value) ;

(axis-number): 1 to the number of controlled axes (number of controlled axes for each path in a multipath system)
(numeric-value): Value specified for the axis specified with its axis number

If an invalid axis number is specified, an alarm PS0331 occurs. If a value is put in a decimal place, the number is rounded off to an integer and the result is treated as the axis number.
You can also specify a variable (local, common, or system variable) for (axis-number). To specify an operation using a variable name in (axis-number), however, enclose the variable name with brackets ([ ]).

1. **AX[1]=100.0;**
   Specifies a value of 100.000 for the first axis.
2. **AX[#500]=200.0;**
   Specifies a value of 200.000 for the axis having the axis number indicated by the value stored in #500.
3. **AX[#500+1]=300.0;**
   Specifies a value of 300.000 for the axis having the axis number obtained by adding 1 to the value stored in #500.
4. **SETVN 500 [ABC];**
   AX[#ABC]=400.0;
   Specifies a value of 400.000 for the axis having the axis number indicated by the value stored in #ABC (#500).
5. **SETVN 500 [ABC];**
   AX[[#ABC]+1]=500.0;
   Specifies a value of 400.000 for the axis having the axis number obtained by adding 1 to the value stored in #ABC (#500).
6. **SETVN 500 [ABC];**
   AX[#ABC+1]=500.0;
   Alarm PS0331 occurs.

- AXNUM function

You can use AXNUM[ ] to obtain an axis number.

AXNUM[(axis-name)];

If an invalid axis name is specified, an alarm PS0332 occurs.

When the number of controlled axes is 3, the name of the first axis is X, that of the second axis is Y, and that of the third axis is Z

1. **#500=AXNUM[X];**
   A value of 1 is stored in #500.
2. **#501=AXNUM[Y];**
   A value of 2 is stored in #501.
3. **#502=AXNUM[Z];**
   A value of 3 is stored in #502.
4. **#503=AXNUM[A];**
   Alarm PS0332 occurs.

Example

Examples when the name of the first axis is X, that of the second axis is Y, and that of the third axis is Z1

```
N10   SETVN 500[AXIS1,AXIS2,AXIS3] ;
N20   [#AXIS1]=AXNUM[X] ;
N30   [#AXIS2]=AXNUM[Y] ;
N40   [#AXIS3]=AXNUM[Z1] ;
N50   G92 AX[#AXIS1]=0 AX[#AXIS2]=0 AX[#AXIS3]=0 ;
N60   G01F1000. ;
N70   AX[#AXIS1]=100.0 AX[#AXIS2]=100.0 AX[#AXIS3]=100.0 ;
N80   G02 AX[#AXIS1]=200. 0 AX[#AXIS1]=200.0 R50.0 ;
N90   M02;
```

Limitation

When the custom macro function is enabled, AX and AXN cannot be used as an extended axis name. AX is assumed to be AX[ ] and AXN is assumed to be AXNUM[ ].

16.5 MACRO STATEMENTS AND NC STATEMENTS

The following blocks are referred to as macro statements:

- Blocks containing an arithmetic or logic operation (=)
- Blocks containing a control statement (such as GOTO, DO, END)
- Blocks containing a macro call command (such as macro calls by G65, G66, G66.1, G67, or other G codes, or by M codes)

Any block other than a macro statement is referred to as an NC statement.

Explanation

- Differences from NC statements

- Even when single block mode is on, the machine does not stop. Note, however, that the machine stops in the single block mode when bit 5 of parameter SBM No. 6000 is 1.
- Macro blocks are not regarded as blocks that involve no movement in the cutter compensation mode.

- NC statements that have the same property as macro statements

- NC statements that include a subprogram call command (such as subprogram calls by M98 or other M codes, or by T codes) and not include other command addresses except an O, file name, N, P, or L address have the same property as macro statements.
- NC statements that include M99 and those do not include other command addresses except an O, file name, N, P, or L address have the same property as macro statements.

16.6 BRANCH AND REPETITION

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

Branch and repetition	GOTO	(unconditional branch)
	IF	(conditional branch: if ..., then...)
	WHILE	(repetition while ...)

16.6.1 Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999999 is specified, an alarm PS0128 occurs. A sequence number can also be specified using an expression.

GOTOn ;	n: Sequence number (1 to 99999999)

Example:

```
GOTO 1;
GOTO #10;
```

⚠ WARNING

Do not specify multiple blocks with the same sequence number in a single program. It is very dangerous to specify such blocks because the destination of a branch from the GOTO statement is undefined.

NOTE

1 A backward branch takes more time as compared with a forward branch.
2 In the destination of GOTOn, that is, the block with sequence number n, the sequence number must appear at the beginning of the block. If the sequence number is not at the beginning of the block, a branch cannot be made.

16.6.2 GOTO Statement Using Stored Sequence Numbers

When the GOTO statement is executed in a custom macro control command, a sequence number search is made for sequence numbers stored at previous execution of the corresponding blocks at a high speed.
As a "sequence number stored at previous execution of the corresponding block," a unique sequence number within the same program or the sequence number of a subprogram call with which the block was executed is stored.
The storage type differs depending on the values of the following parameters.

(1) When bit 1 (MGO) of parameter No. 6000 is set to 1
- Fixed type: Up to 20 sequence numbers stored at execution of the corresponding blocks from the start of operation

(2) When bit 4 (HGO) of parameter No. 6000 is set to 1
- Variable type: Up to 30 sequence numbers stored at execution of the corresponding blocks before execution of the GOTO statement
- History type: Up to 10 sequence numbers stored by a sequence number search previously made using the GOTO statement

The stored sequence numbers are canceled in the following cases:
- Immediately after power-on
- After a reset
- Operation after program registration or editing (including background editing and MDI program editing)

> **⚠ WARNING**
>
> Do not specify multiple blocks with the same sequence number in a single program.
> It is very dangerous to specify the sequence number of the branch destination before and after the GOTO statement and execute the GOTO statement because the branch destination changes according to the values of the parameters as shown below:
>
> | When bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1 | When both bits 1 (MGO) and 4 (HGO) of parameter No. 6000 are set to 0 |
> |---|---|
> | :
N10;
:
GOTO10;
:
N10;

A branch to N10 before the GOTO statement occurs. | :
N10;
:
GOTO10;
:
N10;

A branch to N10 after the GOTO statement occurs. |
>
> When bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1 and the GOTO statement is executed, the sequence number of the branch destination may not be contained in the sequence numbers stored at previous execution of the corresponding blocks. In this case, a branch to the sequence number in a block following the GOTO statement occurs (the destination is the same as when both bits are set to 0).

> **NOTE**
>
> When an external program is read and executed by DNC operation, the executed sequence numbers are not stored.
> When a program registered in memory is executed by a subprogram call, the sequence numbers are stored.

> **⚠ CAUTION**
>
> According to the restrictions on the GOTO statement, no branch to a sequence number within a DO-END loop cannot be made. If a program in which a branch to a sequence number within a loop occurs is executed, operation may differ depending on whether the GOTO statement using stored sequence numbers is used.

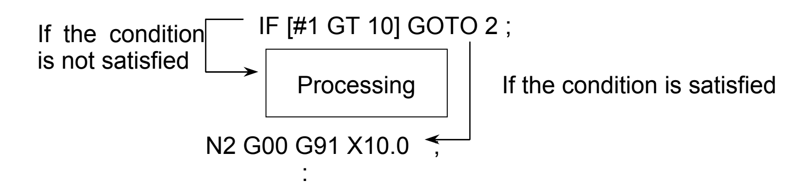
16.6.3 Conditional Branch (IF Statement)

Specify a <conditional expression> after IF.

IF[<conditional expression>]GOTOn

If the specified <conditional expression> is satisfied (true), a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.



IF[<conditional expression>]THEN

If the specified <conditional expression> is satisfied (true), a macro statement specified after THEN is executed.
Only a single macro statement is executed.

If the values of #1 and #2 are the same, 0 is assigned to #3.
IF[#1 EQ #2] THEN#3=0 ;

If the values of #1 and #2 are the same and those of #3 and #4 are also the same, 0 is assigned to #5.
IF[[#1 EQ #2] AND [#3 EQ #4]] THEN#5=0 ;

If the values of #1 and #2 are the same or those of #3 and #4 are the same, 0 is assigned to #5.
IF[[#1 EQ #2] OR [#3 EQ #4]] THEN#5=0 ;

Explanation

- <Conditional expression>

<Conditional expressions> are divided into <simple conditional expressions> and <complex conditional expressions>. In a <simple conditional expression>, a relational operator described in Table 16.6 (a) is specified between two variables or between a variable and constant to be compared. An <expression> can be used instead of a variable. With a <complex conditional expression>, an AND (logical AND), OR (logical OR), or XOR (exclusive OR) operation is performed for the results (true or false) of multiple <simple conditional expressions>.

- Relational operators

Relational operators each consist of two letters and are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value. Note that the equal sign (=) and inequality sign (>, <) cannot be used as a relational operator.

Table 16.6 (a) Relational operators

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≥)
LT	Less than(<)
LE	Less than or equal to(≤)

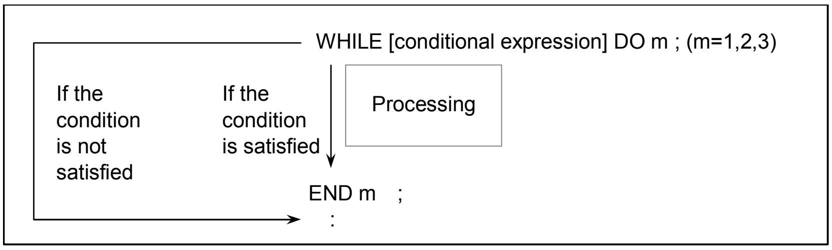
Sample program

The sample program below finds the total of numbers 1 to 10.

```
O9500;
  #1=0;................................ Initial value of the variable to hold the sum
  #2=1;................................ Initial value of the variable as an addend
N1 IF[#2 GT 10] GOTO 2; ... Branch to N2 when the addend is greater than
    10
  #1=#1+#2;........................ Calculation to find the sum
  #2=#2+1;.......................... Next addend
  GOTO 1; .......................... Branch to N1
N2 M30;............................... End of program
```

16.6.4 Repetition (WHILE Statement)

Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If the specified condition is not satisfied, program execution proceeds to the block after END.

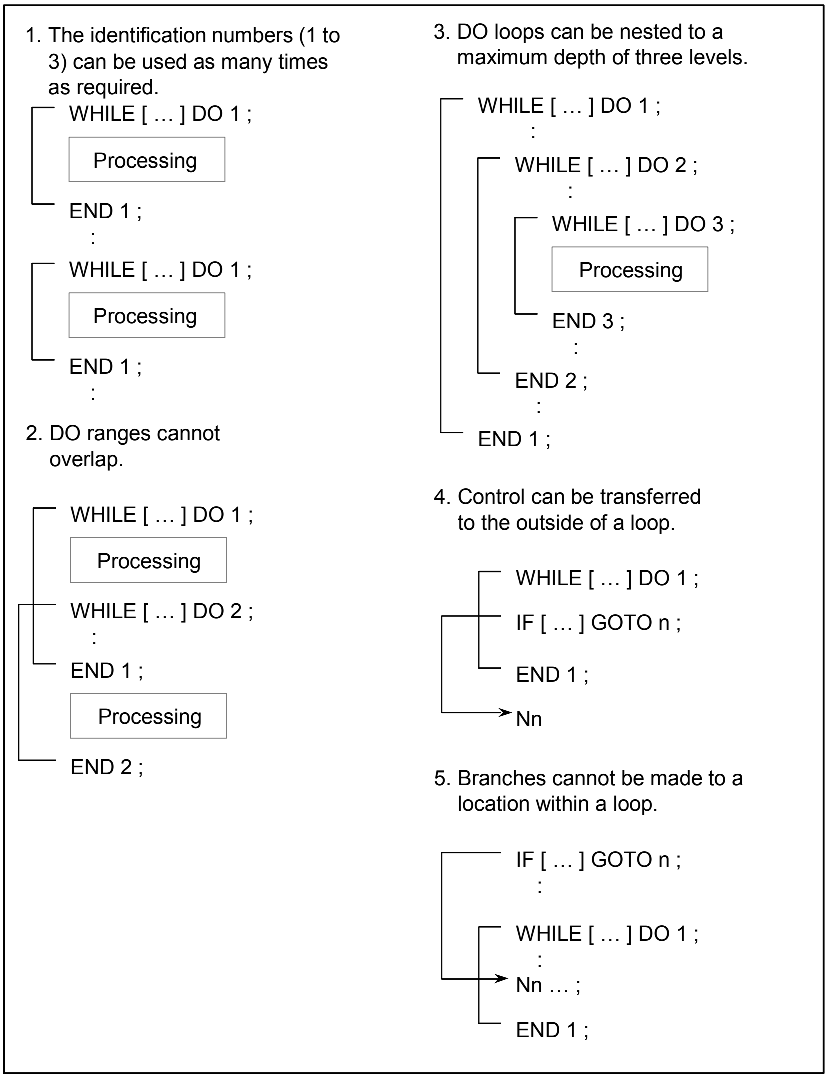


Explanation

While the specified condition is satisfied, the program from DO to END after WHILE is executed. If the specified condition is not satisfied, program execution proceeds to the block after END. The same format as for the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The numbers 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, an alarm PS0126 occurs.

- Nesting

The identification numbers (1 to 3) in a DO-END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), an alarm PS0124 occurs.



Limitation

- Infinite loops

When DO m is specified without specifying the WHILE statement, an infinite loop ranging from DO to END is produced.

- Processing time

When a branch to the sequence number specified in a GOTO statement occurs, the sequence number is searched for. For this reason, processing in the reverse direction takes a longer time than processing in the forward direction. Therefore, in the case of processing in the reverse direction, use the WHILE statement for repetition to reduce processing time.

- Undefined variable

In a conditional expression that uses EQ or NE, a <null> and zero have different effects. In other types of conditional expressions, a <null> is regarded as zero.

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

16.7 MACRO CALL

A macro program can be called using the following methods. The calling methods can roughly be divided into two types: macro calls and subprogram calls.
A macro program can also be called during MDI operation in the same way.

- Macro call
  - Simple call (G65)
  - Modal call (G66, G66.1, G67)
  - Macro call using a G code
  - Macro call using an M code
- Subprogram call
  - Subprogram call using an M code
  - Subprogram call using a T code
  - Subprogram call using an S code
  - Subprogram call using a second auxiliary function code
  - Subprogram call using a specific code

Limitation

- Call nesting

Macro calls can be nested to a depth of up to five levels and subprogram calls can be nested to a depth of up to ten levels; calls can be nested to a depth of up to 15 levels in total.

- Differences between macro calls and subprogram calls

Macro call (G66, G66.1, Ggg, or Mmm) differs from subprogram call (for example M98, Mmm, or Ttt) as described below.

- With a macro call, an argument (data passed to a macro) can be specified. A subprogram call does not have this capability.
- If a macro call block contains another NC command (such as G01 X100.0 G65 Pp), an alarm PS0127 occurs.
- If a subprogram call block contains another NC command (such as G01 X100.0 M98 Pp), the subprogram is called after the command is executed.
- In any macro call block, the machine does not stop in the single block mode.
  If a subprogram call block contains another NC command (such as G01 X100.0 M98 Pp), the machine stops in the single block mode.
- With a macro call, the level of local variables changes. With a subprogram call, the level of local variables does not change. (See "Local variable levels" in Limitation of Subsection 16.7.1.)

- Called program and folders to be searched

The order in which folders are called depends on the method of calling a macro or subprogram.
Folders are searched in sequence and the program found first is called.
For details, see the "Managing Programs" chapter.

16.7.1 Simple Call (G65)

When G65 is specified, the custom macro specified at address P is called. Data (argument) can be passed to the custom macro program.

```
G65 P p L ℓ <argument-specification> ;
```

P : Number of the program to call
ℓ : Repetition count (1 by default)
Argument : Data passed to the macro

```
O0001 ;
   :
G65 P9010 L2 A1.0 B2.0 ;
   :
M30 ;
```

```
O9010 ;
#3=#1+#2 ;
IF [#3 GT 360] GOTO 9 ;
G00 G91 X#3 ;
N9 M99 ;
```

Explanation

- Call

- After G65, specify at address P the program number of the custom macro to call.
- When a number of repetitions is required, specify a number from 1 to 999999999 after address L. When L is omitted, 1 is assumed.
- By using argument specification, values are assigned to corresponding local variables.

- Argument specification

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N, and P once each. Argument specification II uses A, B, and C once each and also uses I, J, and K up to ten times. The type of argument specification is determined automatically according to the letters used.

- Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

- Addresses G, L, N, O, and P cannot be used in arguments.
- Addresses that need not be specified can be omitted. Local variables corresponding to an omitted address are set to null.
- Addresses do not need to be specified alphabetically. They conform to word address format.
  I, J, and K need to be specified alphabetically, however.
  Argument specification I is always used for I, J, and K by setting bit 7 (IJK) of parameter No. 6008 to 1

Example

- When bit 7 (IJK) of parameter No. 6008 is 0, I_J_K_ means that I = #4, J = #5, and K = #6 while K_J_I_ means K = #6, J = #8, and I= #10 because argument specification II is used.
- When bit 7 (IJK) of parameter No. 6008 is 1, K_J_I_ means that I = #4, J = #5, and K = #6, which is the same as with I_J_K_, because argument specification I is used.

- Argument specification II
  Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three-dimensional coordinates as arguments.

Address	Variable number
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11

Address	Variable number
IK_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22

Address	Variable number
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

- Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

NOTE

When bit 7 (IJK) of parameter No. 6008 is 1, argument II cannot be used.

Limitation

- Format

G65 must be specified before any argument.

- Mixture of argument specifications I and II

The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence.

[Example]

G65 A1.0 B2.0 I-3.0 I4.0 D5.0 P1000 ;

(Variables)
#1:1.0
#2:2.0
#3:
#4:-3.0
#5:
#6:
#7:4.0 5.0

When both the I4.0 and D5.0 arguments are commanded for variable #7 in this example, the latter, D5.0, is valid.

- Position of the decimal point

The units used for argument data passed without a decimal point correspond to the least input increment of each address.

> ⚠ **CAUTION**
> The value of an argument passed without a decimal point may vary according to the system configuration of the machine. It is good practice to use decimal points in macro call arguments to maintain program compatibility.

M

When a value is specified with no decimal point, the number of decimal places is determined as follows.

Address	For a non-axis address	For an axis address
D, E, H, M, S, or T	0	
Q or R	α (NOTE 2)	
A, C, I, J, K, X, Y, or Z	α (NOTE 2)	β (NOTE 3)
B, U, V(NOTE 1), or W	0	β (NOTE 3)
Second auxiliary function	γ (NOTE 4)	

Address	Metric input	Inch input
F (G93 mode)	3	
F (G94 mode)	0	2
F (G95 mode)	2 (NOTE 5)	4 (NOTE 5)

NOTE

1 When V is used in a call using a specific code, the number of decimal places is determined according to the setting for the reference axis.

2 α is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the table in NOTE 3.

3 β is determined according to the increment system for the corresponding axis address as listed in the following table.

Increment system	Linear axis (metric input)	Linear axis (inch input)	Rotation axis
IS-A	2	3	2
IS-B	3	4	3
IS-C	4	5	4
IS-D	5	6	5
IS-E	6	7	6

When bit 7 (IPR) of parameter No. 1004 is set to 1, the above values from which 1 is subtracted are used. When the increment system for an axis is IS-A, however, the setting of bit 7 (IPR) of parameter No. 1004 is not effective. When calculator-type decimal notation for each axis is used (bit 0 (ADX) of parameter No. 3455 is set to 1), the number of decimal places is 0. When bit 7 (EAP) of parameter No. 3452 is set to 1, however, calculator-type decimal notation is not effective and the number of decimal places is determined as listed in the above table.

Increment system for the reference axis	AUP (No.3450#0) = 0	AUP(3450#0) = 1			
		AUX (No.3405#0) = 0		AUX (No.3405#0) = 1	
		Metric	Inch	Metric	Inch
IS-A		2		2	3
IS-B		3		3	4
IS-C	0	4		4	5
IS-D		5		5	6
IS-E		6		6	7

5 When bit 1 (FR3) of parameter No. 1405 is 1, the values in the table need to be incremented by 1.

6 When calculator-type decimal notation is used (bit 0 (DPI) of parameter No. 3401 is set to 1), the number of decimal places is 0.

T

When a value is specified with no decimal point, the number of decimal places is determined as follows.

Address	For a non-axis address	For an axis address
H, M, Q, S, or T	0	
D or R	α (NOTE 1)	
A, B, C, I, J, K, U, V, W, X, Y, or Z	α (NOTE 1)	β (NOTE 2)
Second auxiliary function	γ (NOTE 3)	

Address	Metric input	Inch input
E, F (G98 mode)	0 (NOTE 4)	2 (NOTE 4)
E, F (G99 mode)	4	6

NOTE

1 α is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the table in NOTE 2.

2 β is determined according to the increment system for the corresponding axis address as listed in the following table.

Increment system	Linear axis (metric input)	Linear axis (inch input)	Rotation axis
IS-A	2	3	2
IS-B	3	4	3
IS-C	4	5	4
IS-D	5	6	5
IS-E	6	7	6

When bit 7 (IPR) of parameter No. 1004 is set to 1, the above values from which 1 is subtracted are used. When the increment system for an axis is IS-A, however, the setting of bit 7 (IPR) of parameter No. 1004 is not effective.

When calculator-type decimal notation for each axis is used (bit 0 (ADX) of parameter No. 3455 is set to 1), the number of decimal places is 0. When bit 7 (EAP) of parameter No. 3452 is set to 1, however, calculator-type decimal notation is not effective and the number of decimal places is determined as listed in the above table.

NOTE

3 γ is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the following table. (When bit 7 (BDX) of parameter No. 3450 is set to 1, γ is also determined in the same way.)

<table>
<tr><th rowspan="3">Increment system for the reference axis</th><th rowspan="3">AUP (No.3450#0) = 0</th><th colspan="4">AUP(3450#0) = 1</th></tr>
<tr><th colspan="2">AUX (No.3405#0) = 0</th><th colspan="2">AUX (No.3405#0) = 1</th></tr>
<tr><th>Metric</th><th>Inch</th><th>Metric</th><th>Inch</th></tr>
<tr><td>IS-A</td><td rowspan="5">0</td><td colspan="2">2</td><td>2</td><td>3</td></tr>
<tr><td>IS-B</td><td colspan="2">3</td><td>3</td><td>4</td></tr>
<tr><td>IS-C</td><td colspan="2">4</td><td>4</td><td>5</td></tr>
<tr><td>IS-D</td><td colspan="2">5</td><td>5</td><td>6</td></tr>
<tr><td>IS-E</td><td colspan="2">6</td><td>6</td><td>7</td></tr>
</table>

4 When bit 2 (FM3) of parameter No. 1404 is 1, the values in the table need to be incremented by 3.

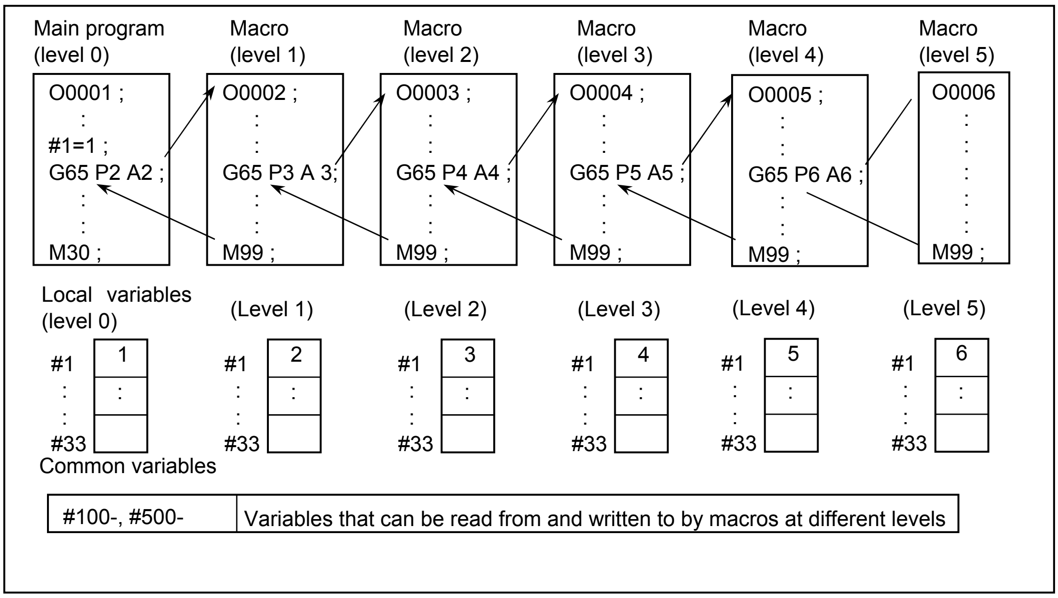
5 When calculator-type decimal notation is used (bit 0 (DPI) of parameter No. 3401 is set to 1), the number of decimal places is 0.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.
A macro program can also be called during MDI operation in the same way.

- Local variable levels

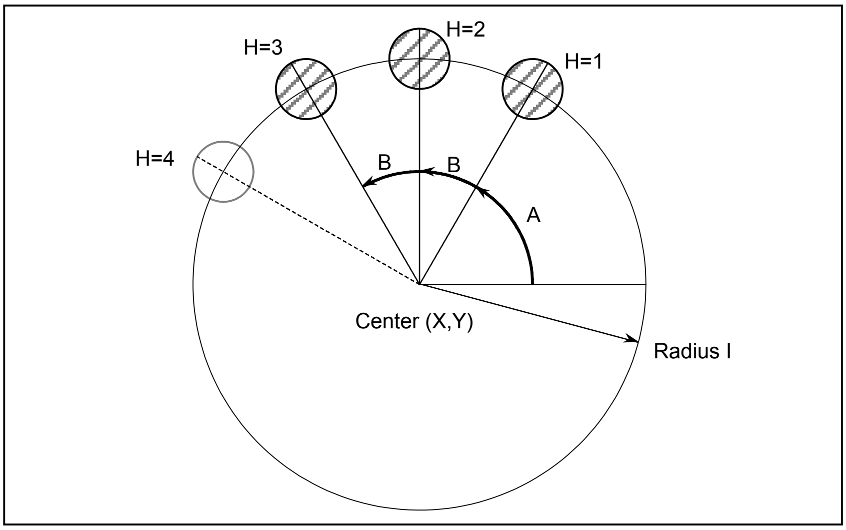
- Local variables from level 0 to 5 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G66, G66.1, Ggg, or Mmm), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC.
- When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.



Sample program (bolt hole circle)

M

A macro is created which drills H holes at intervals of B degrees after a start angle of A degrees along the periphery of a circle with radius I. The center of the circle is (X,Y). Commands can be specified in either the absolute or incremental programming.
To drill in the clockwise direction, specify a negative value for B.



- Calling format

```
G65 P9100 Xx Yy Zz Rr Ff Ii Aa Bb Hh ;
```

X : X coordinate of the center of the circle
(absolute or incremental programming) .... (#24)
Y : Y coordinate of the center of the circle
(absolute or incremental programming) .... (#25)
Z : Hole depth.. (#26)
R : Coordinates of an approach point.. (#18)
F : Cutting feedrate .... (#9)
I : Radius of the circle .... (#4)
A : Drilling start angle .... (#1)
B : Incremental angle
(clockwise when a negative value is specified)....................... (#2)
H : Number of holes .... (#11)

- Program calling a macro program

```
O0002;
G90 G92 X0 Y0 Z100.0;
G65 P9100 X100.0 Y50.0 R30.0 Z-50.0 F500 I100.0 A0 B45.0 H5;
M30;
```

- Macro program (called program)

O9100;	
#3=#4003;	Stores G code of group 3.
G81 Z#26 R#18 F#9 K0; (Note)...	Drilling cycle.
....	Note: L0 can also be used.
IF[#3 EQ 90]GOTO 1;	Branches to N1 in the G90 mode.
#24=#5001+#24;	Calculates the X coordinate of the center.
#25=#5002+#25;	Calculates the Y coordinate of the center.
N1 WHILE[#11 GT 0]DO 1;......	Until the number of remaining holes reaches 0
#5=#24+#4*COS[#1];...............	Calculates a drilling position on the X-axis.
#6=#25+#4*SIN[#1];.................	Calculates a drilling position on the Y-axis.
G90 X#5 Y#6;...........................	Performs drilling after moving to the target position.
#1=#1+#2;	Updates the angle.
#11=#11-1;	Decrements the number of holes.
END 1;	
G#3 G80;..................................	Returns the G code to the original state.
M99;	

Meaning of variables:

#3: Stores the G code of group 3.

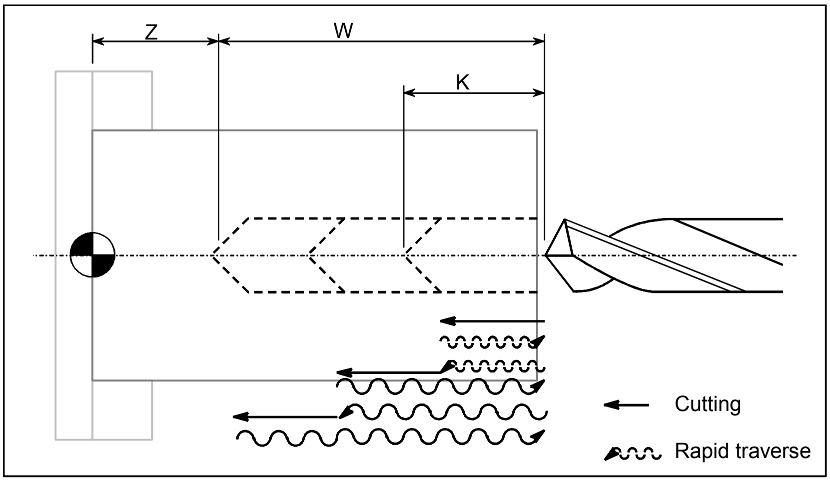
#5: X coordinate of the next hole to drill

#6: Y coordinate of the next hole to drill

Sample program (Drill cycle)

T

Move the tool beforehand along the X- and Z-axes to the position where a drilling cycle starts. Specify Z or W for the depth of a hole, K for the depth of a cut, and F for the cutting feedrate to drill the hole.



- Calling format

G65 P9100 $\left\{ \begin{array}{c} Zz \\ Ww \end{array} \right\}$ Kk Ff ;

Z : Hole depth (absolute programming)
U : Hole depth (incremental programming)
K : Cutting amount per cycle
F : Cutting feedrate

- Program calling a macro program

```
O0002;
G50 X100.0 Z200.0 ;
G00 X0 Z102.0 S1000 M03 ;
G65 P9100 Z50.0 K20.0 F0.3 ;
G00 X100.0 Z200.0 M05 ;
M30
```

- Macro program (called program)

O9100;	
#1=0 ;	Clear the data for the depth of the current hole.
#2=0 ;	Clear the data for the depth of the preceding hole.
IF [#23 NE #0] GOTO 1 ;	If incremental programming, specifies the jump to N1.
IF [#26 EQ #0] GOTO 8 ;	If neither Z nor W is specified, an error occurs.
#23=#5002-#26 ;	Calculates the depth of a hole.
N1 #1=#1+#6 ;	Calculates the depth of the current hole.
IF [#1 LE #23] GOTO 2 ;	Determines whether the hole to be cut is too deep?
#1=#23 ;	Clamps at the depth of the current hole.
N2 G00 W-#2 ;	Moves the tool to the depth of the preceding hole at the cutting feedrate.
G01 W- [#1-#2] F#9 ;	Drills the hole.
G00 W#1 ;	Moves the tool to the drilling start point.
IF [#1 GE #23] GOTO 9 ;	Checks whether drilling is completed.
#2=#1 ;	Stores the depth of the current hole.
N9 M99	
N8 #3000=1 (NOT Z OR U COMMAND)	Issues an alarm.

16.7.2 Modal Call: Call After the Move Command (G66)

Once G66 is issued to specify a modal call a macro is called after a block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.

G66 P p L ℓ <argument-specification> ;

P : Number of the program to call
ℓ : Repetition count (1 by default)
Argument : Data passed to the macro

```
O0001 ;
   :
G66 P9100 L2 A1.0 B2.0 ;
G00 G90 X100.0 ;
Y200.0 ;
X150.0 Y300.0 ;
G67 ;
   :
M30 ;
```

```
O9100 ;
   :
G00 Z-#1 ;
G01 Z-#2 F300 ;
   :
   :
   :
   :
M99 ;
```

Explanation

- Call

- After G66, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 999999999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.
- In the G66 mode, a macro or subprogram can be called.

- Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.

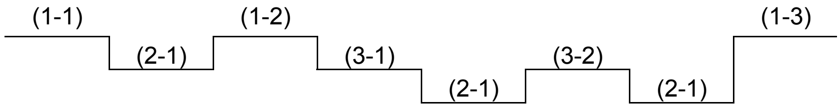
- Modal call nesting

For a single modal call (when G66 is specified only once), each time the move command is executed, the specified macro is called. When nested modal macro calls are specified, the macro at the next higher level is called each time the move command for a macro call is executed.
Macros are called in reverse order in which they are specified. Each time G67 is issued, the macros are canceled one by one in reverse order in which they are specified.

[Example]

```
G66 P9100 ;                   O9100 ;              O9200 ;
X10.0 ;      (1-1)            Z50.0 ;   (2-1)      X60.0 ;   (3-1)
G66 P9200 ;                   M99 ;                Y70.0 ;   (3-2)
X15.0 ;      (1-2)                                 M99;
G67 ;        Cancels P9200.
G67 ;        Cancels P9100.
X-25.0 ;     (1-3)
```

Execution order of the above program (blocks not containing the move command omitted)



* No modal call is performed after (1-3) because the mode is not the macro call mode.

Limitation

- G66 and G67 blocks are specified in pairs in the same program. If a G67 code is specified not in the G66 mode, an alarm PS1100 occurs. Bit 0 (G67) of parameter No. 6000 can be set to 1 to specify that the alarm does not occur in this case.
- In a G66 block, no macros can be called. Local variables (arguments) are set, however.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a auxiliary function that does not involve movement along an axis.
- Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

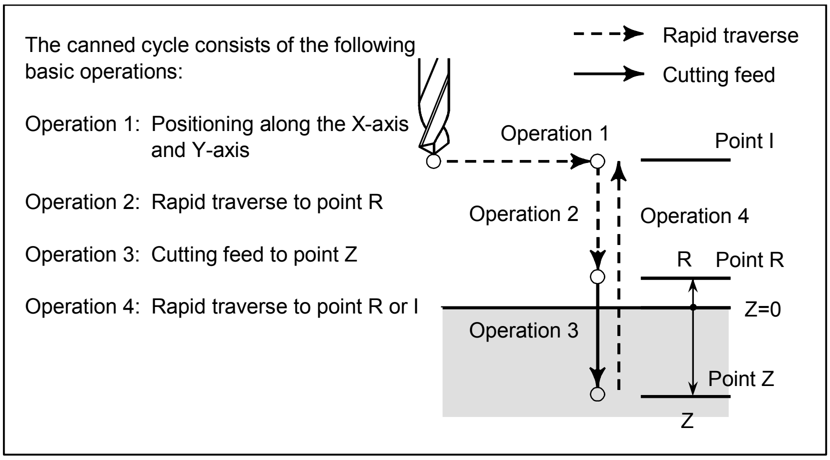
NOTE

If M99 is specified in a block in which a call is performed, it is executed after the call is performed.

Sample program

M

The same operation as the drilling canned cycle G81 is created using a custom macro and the machining program makes a modal macro call. For program simplicity, all drilling data is specified using absolute values.



- Calling format

```
G66 P9110 Zz Rr Ff Ll ;
```

Z : Coordinates of position Z (absolute programming only) ....(#26)
R : Coordinates of position R (absolute programming only).......(#18)
F : Cutting feedrate..(#9)
L : Repetition count

- Program that calls a macro program

```
O0001;
G28 G91 X0 Y0 Z0;
G92 X0 Y0 Z50.0;
G00 G90 X100.0 Y50.0;
G66 P9110 Z-20.0 R5.0 F500;
G90 X20.0 Y20.0;
X50.0;
Y50.0;
X70.0 Y80.0;
G67;
M30;
```

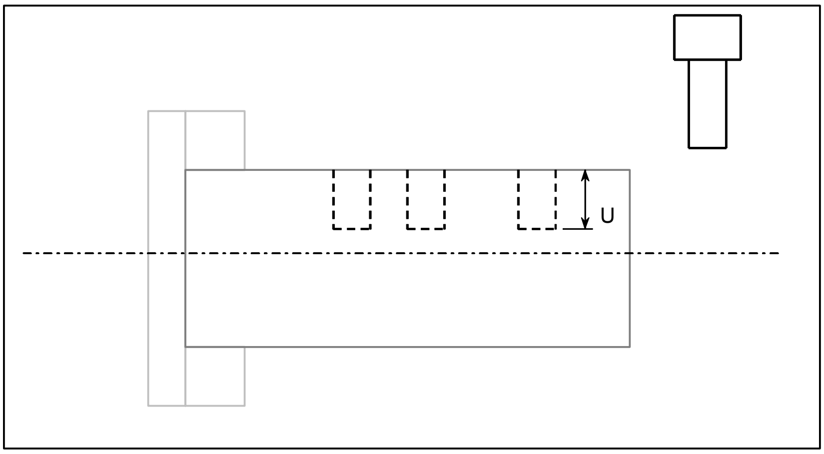
- Macro program (program called)

```
O9110;
  #1=#4001;  .......................Stores G00/G01.
  #3=#4003;.........................Stores G90/G91.
  #4=#4109;.........................Stores the cutting feedrate.
  #5=#5003;.........................Stores the Z coordinate at the start of drilling.
  G00 G90 Z#18; .................Positioning at position R
  G01 Z#26 F#9;..................Cutting feed to position Z
  IF[#4010 EQ 98]GOTO 1; Return to position I
  G00 Z#18;.........................Positioning at position R
  GOTO 2;
N1 G00 Z#5; ........................Positioning at position I
N2 G#1 G#3 F#4; ................Restores modal information.
  M99;
```

Sample program

T

This program makes a groove at a specified position.



- Calling format

G66 P9110 Uu Ff

U : Groove depth (incremental programming)
F : Cutting feed of grooving

- Program that calls a macro program

```
O0003 ;
G50 X100.0 Z200.0 ;
S1000 M03 ;
G66 P9110 U5.0 F0.5 ;
G00 X60.0 Z80.0 ;
Z50.0 ;
Z30.0 ;
G67 ;
G00 X00.0 Z200.0 M05 ;
M30;
```

- Macro program (program called)

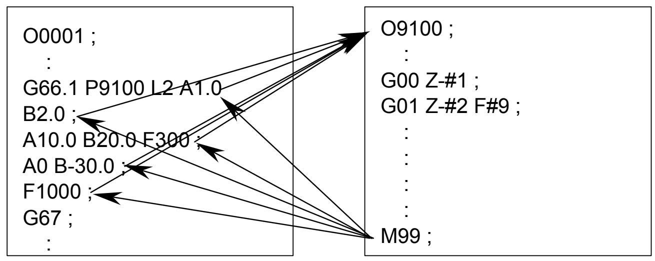
```
O9110 ;
G01 U - #21 F#9 ;....... Cuts the workpiece.
G00 U#21 ; ................. Retracts the tool.
M99 ;
```

16.7.3 Modal Call: Each Block Call (G66.1)

In this macro call mode, the specified macro is unconditionally called for each NC command block. All data other than O, file name, N, and G codes that is specified in each block is not executed and is used as arguments. (The G code in the block in which G66.1 is specified is not used as an argument. Only the last G code specified in subsequent blocks is used as an argument.)
The NC command blocks having O or file name N have the same effect as when G65P is specified in the next position, the other NC command blocks have the same effect as when G65P is specified at the beginning.

G66.1 P p L ℓ <argument-specification> ;

P : Number of the program to call
ℓ : Repetition count (1 by default)
Argument : Data passed to the macro



[Example]
In the G66.1 P100 ; mode,
N001 G01 G91 X100 Y200 D1 R1000 ;
is the same as
N001 G65 P100 G01 G91 X100 Y200 D1 R1000 ;

Explanation

- Call

- After G66.1, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 999999999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.
- In the G66.1 mode, a macro or subprogram can be called.

- Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.

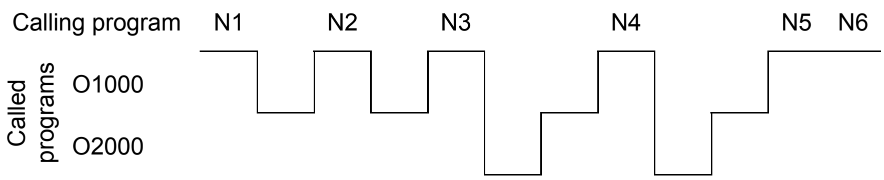
- Modal call nesting

For a single modal call (when G66.1 is specified only once), the specified macro is called for each NC command block. When nested modal macro calls are specified, the macro at the next higher level is also called in a block within a called macro in which an NC command is specified.
Macros are called in reverse order in which they are specified. Each time G67 is issued, the macros are canceled one by one in reverse order in which they are specified.

[Example] Axis specification switching

```
N1 G66.1 P1000 X10.0 ;           → Calls O1000 and executes Y10.0.
N2 X20.0 ;                       → Calls O1000 and executes Y20.0.
N3 G66.1 P2000 Y10.0 Z20.0 ;     → Calls O2000 and executes Y20.0
                                   Z10.0.  Then, calls O1000 and
                                   executes X20.0 Z10.0.
N4 X10.0 Y20.0 Z30.0 ;           → Calls O2000 and executes X10.0
                                   Y30.0 Z20.0.  Then, calls O1000 and
                                   executes X30.0 Y10.0 Z20.0.
N5 G67 ;                         → Cancels P2000.
N6 G67 ;                         → Cancels P1000.


O1000 X#25 Y#24 Z#26 ;          (X-Y switching)
M99 ;


O2000 X#24 Y#26 Z#25 ;          (Y-Z switching)
M99 ;
```

Execution order of the above program (blocks not containing the move command omitted)



In N1 and N2 blocks, O1000 is called and the X and Y specifications are made to change places.
In N3 and N4 blocks, O2000 is called first and the Y and Z specifications are made to change places. For the switched specification, O1000 is called and the X and Y specifications are made to change places. Therefore, the X, Y, and Z specifications are switched to the Z, X, and Y specifications.

Limitation

- G66.1 and G67 blocks are specified in pairs in the same program. If a G67 code is specified not in the G66.1 mode, an alarm PS1100 occurs. Bit 0 (G67) of parameter No. 6000 can be set to 1 to specify that the alarm does not occur in this case.
- G66.1 block
  - (a) In a G66.1 block, a macro is called.
  - (b) The correspondence between addresses specified as arguments and variables is the same as for simple calls.

- Block following a G66.1 block in which a call is performed (not including G66.1 blocks)
  - (a) Addresses G, P, and L can also be used as arguments. Address G corresponds to #10; address L to #12; address P to #16. However, the restrictions on the input format of normal NC commands are put on the data. For example, ;G1000. P0.12 L-4 cannot be specified.
  - (b) When multiple G codes are specified, only the last G code is used as an argument. O, file name, and N codes and G codes not in group 00 are passed to the next and subsequent blocks.

NOTE

1 In a block in which only an O number, file name, sequence number, EOB, macro statement, or M99 command is specified, a macro is not called for each block.

2 In each block, when an address other than O, file name, or N is specified, it is assumed to be an NC command and a macro is called for each block. When N is specified following an address other than O, file name, or N, it is used as an argument. In this case, N corresponds to variable #14 and the number of decimal places is 0.

3 If M99 is specified in a block in which a macro is called, it is executed after the call is performed.

16.7.4 Macro Call Using a G Code

By setting a G code number used to call a macro program in a parameter, the macro program can be called in the same way as for a simple call (G65).

```
O0001 ;                          O9010 ;
     :                                :
G81 X10.0 Y20.0 Z-10.0 ;              :
     :                                :
M30 ;                            N9 M99 ;

Parameter   No.6050=81
```

Explanation

By setting a G code number from -9999 to 9999 used to call a custom macro program (O9010 to O9019) in the corresponding parameter (No.6050 to No.6059), the macro program can be called in the same way as with G65. To call custom macro program O9040 to O9049 using a G code with a decimal point, set bit 0 (DPG) of parameter No. 6007 to 1 and set the G code number in the corresponding parameter (Nos. 6060 to 6069).
The number of decimal places of a G code is 1. Set the value obtained by multiplying a desired G code number by 10 in the corresponding parameter.

[Example] When parameter No. 6060 is set to 234, custom macro program O9040 is called using G23.4.

When a negative G code is set, a modal call is performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.
For example, when a parameter is set so that macro program O9010 can be called with G81, a user-specific cycle created using a custom macro can be called without modifying the machining program.

- Correspondence between parameter numbers and program numbers

G code with no decimal point		G code with a decimal point	
Parameter number	Program number	Parameter number	Program number
6050	O9010	6060	O9040
6051	O9011	6061	O9041
6052	O9012	6062	O9042
6053	O9013	6063	O9043
6054	O9014	6064	O9044
6055	O9015	6065	O9045
6056	O9016	6066	O9046
6057	O9017	6067	O9047
6058	O9018	6068	O9048
6059	O9019	6069	O9049

- Repetition

As with a simple call, a number of repetitions from 1 to 999999999 can be specified at address L.

- Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitation

- Nesting of calls using G codes

- To call another program in a program called using a G code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using an M, T, or S code, second auxiliary function, or specific code can be performed in a program called using a G code.

16.7.5 Macro Call Using a G Code (Specification of Multiple Definitions)

By setting the starting G code number used to call a macro program, the number of the starting program to be called, and the number of definitions, macro calls using multiple G codes can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6040 can be called using as many G codes as the number specified in parameter No. 6040. The numeric value set in parameter No. 6038 indicates the starting G code number and the program number set in parameter No. 6039 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6040.
When a negative G code is set in parameter No. 6038, modal calls are performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.
The number of repetitions and argument specification are set in the same way as with a macro call using a G code.

[Example]

Set parameter No. 6038 to 900, parameter No. 6039 to 10000000, and parameter No. 6040 to 100.

G900 → O10000000
G901 → O10000001
G902 → O10000002
:
G999 → O10000099

Custom macro calls (simple calls) for 100 combinations are defined as shown above. When parameter No. 6038 is changed to -900, custom macro calls (modal calls) for the same combinations are defined.

NOTE

1 The calls defined by this setting become all invalid in the following cases:
<1> A value outside the valid data range is set in one of the above parameters.
<2> (No. 6039 + No. 6040 - 1) > 99999999
2 Simple and modal calls cannot be mixed in the specification.
3 If the G code set in parameter Nos. 6050 to 6059 to call the corresponding macro program is within the G code range for calling programs using multiple G codes, the macro program corresponding to the G code set in parameter Nos. 6050 to 6059 is called.

16.7.6 Macro Call Using a G Code with a Decimal Point (Specification of Multiple Definitions)

When bit 0 (DPG) of parameter No. 6007, by setting the starting G code number with a decimal point used to call a macro program, the number of the starting program to be called, and the number of definitions, multiple macro calls using multiple G codes with a decimal point can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6043 can be called using as many G codes with a decimal point as the number specified in parameter No. 6043. The numeric value set in parameter No. 6041 indicates the starting G code number with a decimal point and the program number set in parameter No. 6042 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6043.
When a negative G code is set in parameter No. 6041, modal calls are performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.
The number of repetitions and argument specification are set in the same way as with a macro call using a G code.
[Example]

Set parameter No. 6041 to 900, parameter No. 6042 to 2000, and parameter No. 6043 to 100.

G90.0 → O2000
G90.1 → O2001
G90.2 → O2002
:
G99.9 → O2099

Custom macro calls (simple calls) for 100 combinations are defined as shown above. When parameter No. 6041 is changed to -900, custom macro calls (modal calls) for the same combinations are defined.

NOTE

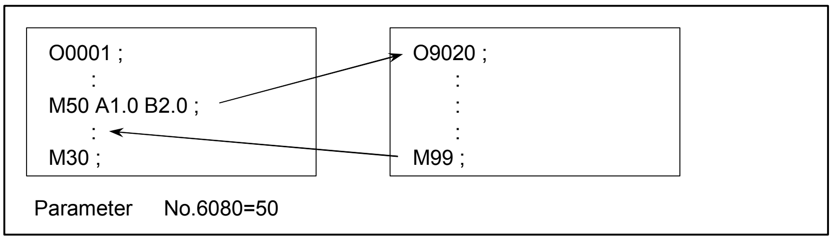
1 The calls defined by this setting become all invalid in the following cases:
<1> A value outside the valid data range is set in one of the above parameters.
<2> (No. 6042 + No. 6043 - 1) > 99999999
<3> Bit 0 (DPG) of parameter No. 6007 is set to 0 (this setting invalidates a macro call using a G code with a decimal point).

2 Simple and modal calls cannot be mixed in the specification.

3 If the G code set in parameter Nos. 6060 to 6069 to call the corresponding macro program is within the G code range for calling programs using multiple G codes, the macro program corresponding to the G code set in parameter Nos. 6060 to 6069 is called.

16.7.7 Macro Call Using an M Code

By setting an M code number used to call a macro program in a parameter, the macro program can be called in the same way as with a simple call (G65).



Explanation

By setting an M code number from 3 to 99999999 used to call custom macro program O9020 to O9029 in the corresponding parameter (Nos. 6080 to 6089), the macro program can be called in the same way as with G65.

- Correspondence between parameter numbers and program numbers

Parameter number	Corresponding program number
6080	O9020
6081	O9021
6082	O9022
6083	O9023
6084	O9024
6085	O9025
6086	O9026
6087	O9027
6088	O9028
6089	O9029

Example)

When parameter No. 6080 is set to 990, O9020 is called using M990.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitation

- An M code used to call a macro program must be specified at the start of a block.
- To call another program in a program called using an M code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.

16.7.8 Macro Call Using an M Code (Specification of Multiple Definitions)

By setting the starting M code number used to call a macro program, the number of the starting program to be called, and the number of definitions, macro calls using multiple M codes can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6049 can be called using as many M codes as the number specified in parameter No. 6049. The numeric value set in parameter No. 6047 indicates the starting M code number and the program number set in parameter No. 6048 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6049.
The number of repetitions and argument specification are set in the same way as with a macro call using a M code.

[Example]

Set parameter No. 6047 to 9000, parameter No. 6048 to 4000, and parameter No. 6049 to 100.

M90000000 → O4000
M90000001 → O4001
M90000002 → O4002
:
M90000099 → O4099

Custom macro calls (simple calls) for 100 combinations are defined as shown above.

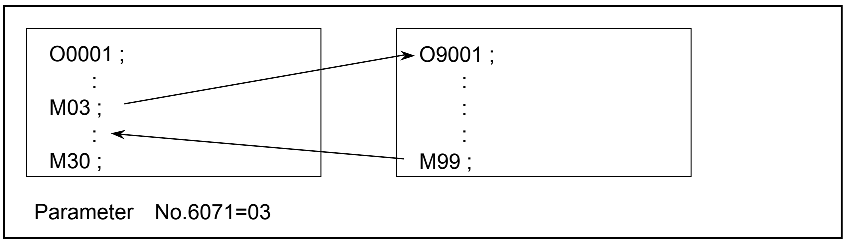
NOTE

1 The calls defined by this setting become all invalid in the following cases:
<1> A value outside the valid data range is set in one of the above parameters.
<2> (No. 6048 + No. 6049 - 1) > 99999999

2 If the M code set in parameter Nos. 6080 to 6089 to call the corresponding macro program is within the M code range for calling programs using multiple M codes, the macro program corresponding to the M code set in parameter Nos. 6080 to 6089 is called.

16.7.9 Subprogram Call Using an M Code

By setting an M code number used to call a subprogram (macro program) in a parameter, the macro program can be called in the same way as with a subprogram call (M98).



Explanation

By setting an M code number from 3 to 99999999 used to call subprogram O9001 to O9009 in the corresponding parameter (No. 6071 to 6079), the subprogram can be called in the same way as with M98.

- Correspondence between parameter numbers and program numbers

Parameter number	Program number
6071	O9001
6072	O9002
6073	O9003
6074	O9004
6075	O9005
6076	O9006
6077	O9007
6078	O9008
6079	O9009

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

Argument specification is not allowed.

- M code

An M code in a macro program that has been called is treated as an ordinary M code.

Limitation

- To call another program in a program called using an M code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.

16.7.10 Subprogram Call Using an M Code (Specification of Multiple Definitions)

By setting the starting M code number used to call a subprogram, the number of the starting subprogram to be called, and the number of definitions, subprogram calls using multiple M codes can be defined.

Explanation

As many subprograms as the number specified in parameter No. 6046 can be called using as many M codes as the number specified in parameter No. 6046. The numeric value set in parameter No. 6044 indicates the starting M code number and the numeric value set in parameter No. 6045 indicates the starting subprogram number. To disable this type of call, set 0 in parameter No. 6046.

[Example]

Set parameter No. 6044 to 80000000, parameter No. 6045 to 30000000, and parameter No. 6046 to 100.

M80000000 → O3000
M80000001 → O3001
M80000002 → O3002
:
M80000099 → O3099

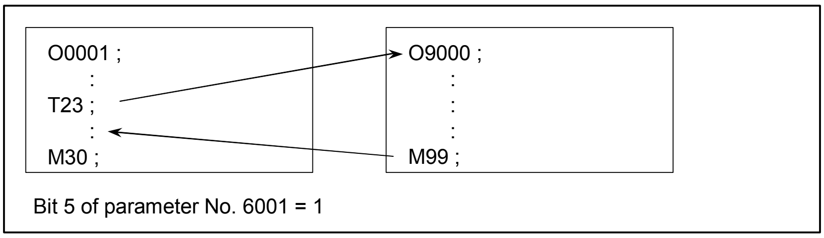
Subprogram calls for 100 combinations are defined as shown above.

NOTE

1 The calls defined by this setting become all invalid in the following cases:
  <1> A value outside the valid data range is set in one of the above parameters.
  <2> (No. 6045 + No. 6046 - 1) > 99999999
2 If the M code set in parameter Nos. 6071 to 6079 to call the corresponding subprogram is within the M code range for calling subprograms using multiple M codes, the subprogram corresponding to the M code set in parameter Nos. 6071 to 6079 is called.

16.7.11 Subprogram Calls Using a T Code

By enabling subprograms to be called with a T code in a parameter, a subprogram can be called each time the T code is specified in the machining program.



Explanation

- Call

By setting bit 5 (TCS) of parameter No. 6001 to 1, subprogram O9000 can be called each time a T code is specified in a machining program. A T code specified in a machining program is assigned to common variable #149.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

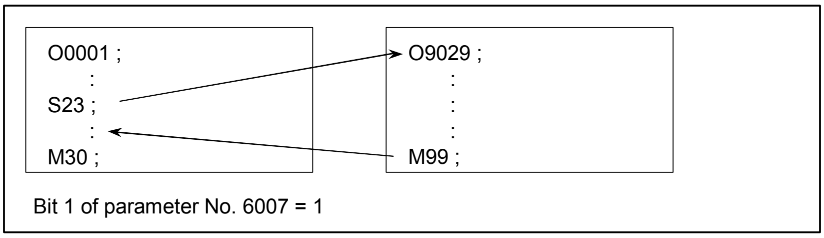
Argument specification is not allowed.

Limitation

- To call another program in a program called using a T code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a T code.

16.7.12 Subprogram Calls Using an S Code

By enabling subprograms to be called with an S code in a parameter, a subprogram can be called each time the S code is specified in the machining program.



Explanation

- Call

By setting bit 1 (SCS) of parameter No. 6007 to 1, subprogram O9029 can be called each time a S code is specified in a machining program. An S code specified in a machining program is assigned to common variable #147.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

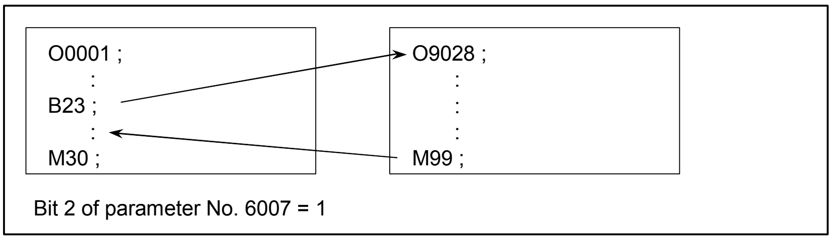
Argument specification is not allowed.

Limitation

- To call another program in a program called using an S code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an S code.

16.7.13 Subprogram Calls Using a Secondary Auxiliary Function

By enabling subprograms to be called with a secondary auxiliary function in a parameter, a subprogram can be called each time the secondary auxiliary function is specified in the machining program.



Explanation

- Call

By setting bit 2 (BCS) of parameter No. 6007 to 1, subprogram O9028 can be called each time a secondary auxiliary function code is specified in a machining program. A secondary auxiliary function specified in a machining program is assigned to common variable #146.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

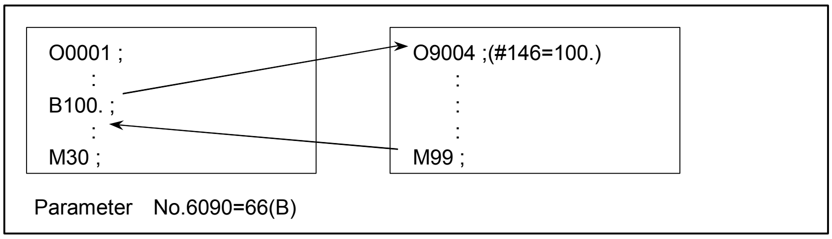
Argument specification is not allowed.

Limitation

- To call another program in a program called using a second auxiliary function, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a second auxiliary function.

16.7.14 Subprogram Call Using a Specific Address

By enabling subprograms to be called with a specific address in a parameter, a subprogram can be called each time the specific address is specified in the machining program.



Explanation

- Call

By setting the code (ASCII code converted to decimal) corresponding to a specific address in parameter No. 6090 or No. 6091, the custom macro program, O9004 or O9005, corresponding to each parameter can be called when the specific address is specified in a machining program. The code value corresponding to a specific address specified in a machining program is assigned to the common variables (#146, #147). The table below indicates the addresses that can be set.

M

Address	Parameter setting
A	65
B	66
D	68
F	70
H	72
I	73
J	74
K	75
L	76
M	77
P	80
Q	81
R	82
S	83
T	84
V	86
X	88
Y	89
Z	90

NOTE

When address L is set, the number of repetitions cannot be set.

T

Address	Parameter setting
A	65
B	66
F	70
H	72
I	73
J	74
K	75
L	76
M	77
P	80
Q	81
R	82
S	83
T	84

NOTE

When address L is set, the number of repetitions cannot be set.

- Correspondence between parameter numbers and program numbers and between the parameter numbers and common variables

Parameter number	Program number	Common variable
6090	O9004	#146
6091	O9005	#147

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

Argument specification is not allowed.

Limitation

- To call another program in a program called using a specific code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a specific code.

Sample program

By using the subprogram call function that uses M codes, the cumulative usage time of each tool is measured.

Conditions

- The cumulative usage time of each of tools T01 to T05 is measured.
  No measurement is made for tools with numbers greater than T05.

- The following variables are used to store the tool numbers and measured times:

#501	Cumulative usage time of tool number 1
#502	Cumulative usage time of tool number 2
#503	Cumulative usage time of tool number 3
#504	Cumulative usage time of tool number 4
#505	Cumulative usage time of tool number 5

- Usage time starts being counted when the M03 command is specified and stops when M05 is specified. System variable #3002 is used to measure the time during which the cycle start lamp is on. The time during which the machine is stopped by feed hold and single block stop operation is not counted, but the time used to change tools and pallets is included.

Operation check

- Parameter setting

Set 3 in parameter No.6071, and set 5 in parameter No.6072.

- Variable value setting

Set 0 in variables #501 to #505.

- Program that calls a macro program

```
O0001;
T01 M06;
M03;
    :
M05; ........................................ Changes #501.
T02 M06;
M03;
    :
M05; ........................................ Changes #502.
T03 M06;
M03;
    :
M05; ........................................ Changes #503.
T04 M06;
M03;
    :
M05; ........................................ Changes #504.
T05 M06;
M03;
    :
M05; ........................................ Changes #505.
M30;
```

- Macro program (program called)

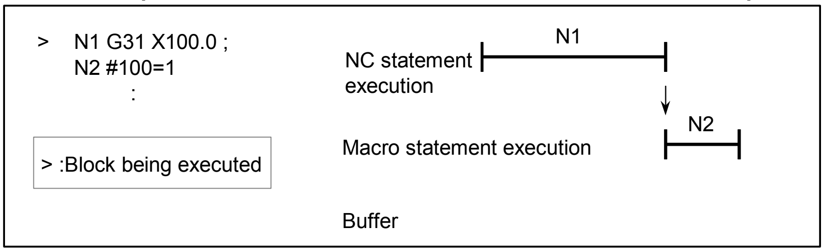
```
O9001(M03); ..................................................Macro to start counting
  M01;
  IF[#4120 EQ 0]GOTO 9; ..............................No tool specified
  IF[#4120 GT 5]GOTO 9;...............................Out-of-range tool number
  #3002=0;......................................................Clears the timer.
N9 M03;.........................................................Rotates the spindle in the
  M99;                                                          forward direction.

O9002(M05); ..................................................Macro to end counting
  M01;
  IF[#4120 EQ 0]GOTO 9; ..............................No tool specified
  IF[#4120 GT 5]GOTO 9;...............................Out-of-range tool number
  #[500+#4120]=#3002+#[500+#4120]; ..........Calculates cumulative time.

N9 M05;.........................................................Stops the spindle.
  M99;
```

16.8 PROCESSING MACRO STATEMENTS

For smooth machining, the CNC prereads the NC statement to be performed next. This operation is referred to as buffering. For example, many NC statements are buffered during acceleration/ deceleration before interpolation.
In the cutter compensation mode (G41 or G42), the CNC prereads the NC statements at least three blocks ahead to find intersections even if acceleration/deceleration before interpolation is not applied.
Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Therefore, the timing of the macro statement execution is not always the specified order.
At the blocks containing M00, M01, M02 or M30, blocks containing M-codes for which buffering is suppressed by setting parameter (No.3411 to 3420 and No.3421 to 3432), and blocks containing prevention buffering G codes such as G31, the CNC stops to preread the NC statement after that. Then, the stop of the macro statement execution is guaranteed until such M codes or G codes complete its execution.

Explanation

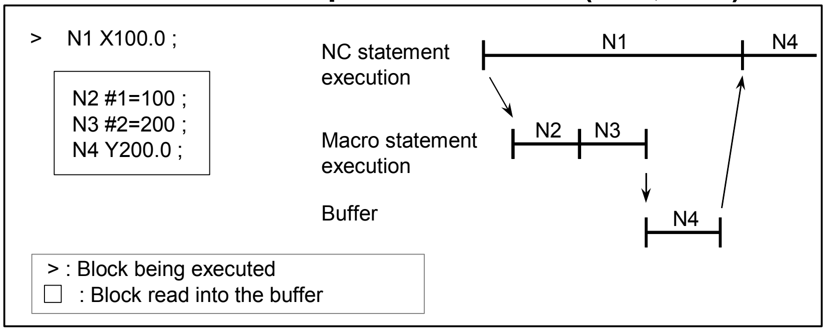
- When the next block is not buffered (M codes that are not buffered, G31, etc.)



⚠CAUTION

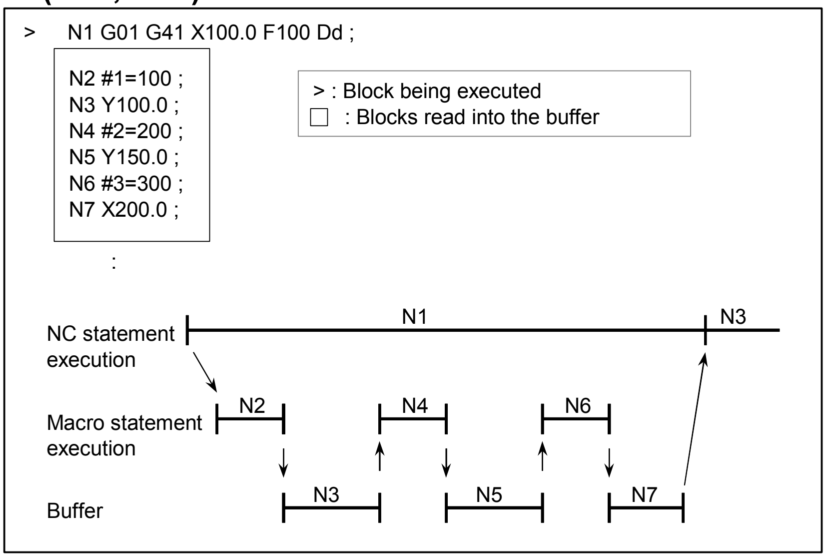
In case that you need to execute the macro statement after completing the block just before the macro statement, specify M code or G code that is not buffered just before the macro statement. Specially, in case of reading/writing the system variables to control signals, coordinates, offset value, etc., it may different system variable data by the timing of the NC statement execution. To avoid this phenomenon, specify such M codes or G codes before the macro statement, if necessary.

- Buffering the next block in other than cutter compensation mode (G41, G42)



When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- In cutter compensation mode (G41, G42)



When N1 is being executed, the NC statements in the next three blocks (up to N7) are read into the buffer. The macro statements (N1, N4, N6) between N1 and N7 are processed during execution of N1.

16.9 REGISTERING CUSTOM MACRO PROGRAMS

Custom macro programs are similar to subprograms. They can be registered and edited in the same way as subprograms. The storage capacity is determined by the total length of tape used to store both custom macros and subprograms.

16.10 CODES AND RESERVED WORDS USED IN CUSTOM MACROS

In addition to the codes used in ordinary programs, the following codes are used in custom macro programs.

Explanation

- Codes

(1) When the ISO code is used or when bit 4 (ISO) of parameter No. 6008 is set to 0
(The codes are represented in hexadecimal.)

Meaning	Code
*	0AAh
=	0BDh
#	0A3h
[	0DBh
]	0DDh
?	03Fh
@	0C0h
&	0A6h
_	05Fh
O	0CFh

(2) When the EIA code is used or when the ISO code is used with bit 4 (ISO) of parameter No. 6008 set to 1

Meaning	Code
*	Code set in parameter No. 6010
=	Code set in parameter No. 6011
#	Code set in parameter No. 6012
[	Code set in parameter No. 6013
]	Code set in parameter No. 6014
?	Code set in parameter No. 6015
@	Code set in parameter No. 6016
&	Code set in parameter No. 6017
_	Code set in parameter No. 6018

For O, the same code as for O indicating a program number is used. Set a hole pattern for each of *, =, #, [,], ?, @, &, and _ in the ISO or EIA code in the corresponding parameter (Nos. 6010 to 6018).
The code 00h cannot be used. The code indicating an alphabetic character can be used for the code indicating a character listed above, but the code can be no longer used to indicate the original character.

- Reserved words

The following reserved words are used in custom macros:

AND, OR, XOR, MOD, EQ, NE, GT, LT, GE, LE,
SIN, COS, TAN, ASIN, ACOS, ATAN, ATN, SQRT, SQR, ABS, BIN, BCD, ROUND, RND, FIX, FUP, LN, EXP, POW, ADP, IF, GOTO, WHILE, DO, END, BPRNT, DPRNT, POPEN, PCLOS, SETVN

System variable (constant) names and registered common variable names are also used as reserved words.

16.11 EXTERNAL OUTPUT COMMANDS

In addition to the standard custom macro commands, the following macro commands are available. They are referred to as external output commands.

- BPRNT
- DPRNT
- POPEN
- PCLOS

These commands are provided to output variable values and characters through the reader/puncher interface.

Explanation

Specify these commands in the following order:

Open command: POPEN
Before specifying a sequence of data output commands, specify this command to establish a connection to an external input/output device.
Data output command: BPRNT or DPRNT
Specify necessary data output.
Close command: PCLOS
When all data output commands have completed, specify PCLOS to release a connection to an external input/output device.

- Open command POPEN

The POPEN command establishes a connection to an external input/output device. It must be specified before a sequence of data output commands. The CNC outputs a DC2 control code.

- Data output command BPRNT

The BPRNT command outputs characters and variable values in binary.

BPRNT [a #b [c] …]

c: Number of significant decimal places
#b: Variable
a: Character

(i) Specified characters are converted to the codes according to the setting data (ISO) that is output at that time.
Specifiable characters are as follows:
- Letters (A to Z)
- Numbers
- Special characters (*, /, +, -, ?, @, &, _)

NOTE
1 An asterisk (*) is output by a space code.
2 When using ?, @, &, and/or _, use the ISO code as the punch code (setting data (ISO) = 1).

(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2-word (32-bit) data, including the decimal digits. It is output as binary data starting from the highest byte.
(iii) When specified data has been output, an EOB code is output according to the setting code (ISO).
(iv) <Null> variables are regarded as 0.

Example
BPRNT [C** X#100 [3] Y#101 [3] M#10 [0]]
Variable value
#100=0.40956
#101=-1638.4
#10=12.34
are output as follows:

```
C3 A0 A0 D8 00 00 01 9A 59 FF E7 00 00 4D 00 00 00 0C 0A
↓                                                     ↓
C  sp sp  X0000019A    YFFE70000    M0000000C    LF
   ( * )  (410)        (-1638400)   (12)         ( ; )
```

- Data output command DPRNT

DPRNT [a #b [c d] …]

- a: Character
- #b: Variable
- c: Number of significant digits in the integer part
- d: Number of significant decimal places

The DPRNT command outputs characters and each digit in the value of a variable according to the code set in the settings (ISO).

(i) For an explanation of the DPRNT command, see Items (i), (iii), and (iv) for the BPRNT command.

(ii) When outputting a variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in brackets.
For the value of a variable, as many codes as the specified number of digits are output according to the settings one by one, starting with the highest digit. The decimal point is also output using the set code.
Each variable must be a numeric value consisting of up to nine digits. When high-order digits are zeros, these zeros are not output if PRT (bit1 of parameter 6001) is 1. If parameter PRT is 0, a space code is output each time a zero is encountered.
When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output.
When PRT (bit 1 of parameter 6001) is 0, a space code is output to indicate a positive number instead of +; if parameter PRT is 1, no code is output.

Example
DPRNT [X#2 [53] Y#5 [53] T#30 [20]]
Variable value
#2=128.47398
#5=-91.2
#30=123.456
are output as follows:

(1) Parameter PRT (No.6001#1) = 0

D8 A0 A0 A0 B1 B2 B8 2E B4 B7 B4 59 2D A0 A0 A0 39 B1 2E B2 30 30 D4 A0 B2 33 0A

X sp sp sp 128.474 Y- sp sp sp 91.200 T sp 023 LF

(2) Parameter PRT (No.6001#1) = 1

D8 B1 B2 B8 2E B4 B7 B4 59 2D 39 B1 2E B2 30 30 D4 A0 B2 33 0A

X128.474 Y-91.200 T023 LF

- Close command PCLOS

The PCLOS command releases a connection to an external input/output device. Specify this command when all data output commands have terminated. DC4 control code is output from the CNC.

- Required setting

Specify the specification number of an input/output device to be used for the I/O device specification number.
According to the above settings, set data items (such as the baud rate) for the reader/puncher interface.
Never specify the output device FANUC Cassette or Floppy for punching.
When specifying a DPRNT command to output data, specify whether leading zeros are output as spaces (by setting PRT (bit 1 of parameter 6001) to 1 or 0).
To indicate the end of a line of data in ISO code, specify whether to use only an LF (CRO, of bit 4 of parameter 6001 is 0) or an LF and CR (CRO of bit 4 of parameter 6001 is 1).

NOTE

1 It is not necessary to always specify the open command (POPEN), data output command (BPRNT, DPRNT), and close command (PCLOS) together. Once an open command is specified at the beginning of a program, it does not need to be specified again except after a close command was specified.
2 Be sure to specify open commands and close commands in pairs. Specify the close command at the end of the program. However, do not specify a close command if no open command has been specified.
3 When a reset operation is performed while commands are being output by a data output command, output is stopped and subsequent data is erased. Therefore, when a reset operation is performed by a code such as M30 at the end of a program that performs data output, specify a close command at the end of the program so that processing such as M30 is not performed until all data is output.

16.12 RESTRICTIONS

- Sequence number search

A custom macro program cannot be searched for a sequence number.

- Single block

Even while a macro program is being executed, blocks can be stopped in the single block mode.
A block containing a macro call command (G66, G66.1, Ggg, Mmm, or G67) does not stop even when the single block mode is on.
Whether blocks containing arithmetic and logic operation commands and control commands are stopped depends on the settings of bits 5 (SBM) and 7 (SBV) of parameter No. 6000 as shown in the following table.

		Bit 5 (SBM) of parameter No. 6000	
		0	**1**
Bit 7 (SBV) of parameter No. 6000	**0**	Not stopped when the single block mode is on.	Can be stopped in the single block mode. (Variable #3003 cannot be used to disable single block stop. Single block stop is always enabled.)
	1	Can be stopped in the single bock mode. (Variable #3003 can be used to enable or disable single block stop.)	

Note that when a single block stop occurs at a macro statement in cutter compensation mode, the statement is assumed to be a block that does not involve movement, and proper compensation cannot be performed in some cases. (Strictly speaking, the block is regarded as specifying a movement with a travel distance 0.)

- Optional block skip

A / appearing in the middle of an <expression> (enclosed in brackets [ ] on the right-hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specifier for an optional block skip code.

- Operation in EDIT mode

By setting NE8 (bit 0 of parameter 3202) and NE9 (bit 4 of parameter 3202) to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 8000 to 8999 and 9000 to 9999. This prevents registered custom macro programs and subprograms from being destroyed by accident. When the entire memory is cleared, the contents of memory such as custom macro programs are deleted.

- Reset

With a reset operation, local variables and common variables #100 to #199 are cleared to null values. They can be prevented from clearing by setting CCV (bit 6 of parameter 6001). System variables #100 to #199 are not cleared.
A reset operation clears any called states of custom macro programs and subprograms, and any DO states, and returns control to the main program.

- Display of the PROGRAM RESTART

As with M98, the M and T codes used for subprogram calls are not displayed.

- Feed hold

When a feed hold is enabled during execution of a macro statement, the machine stops after execution of the macro statement. The machine also stops when a reset or alarm occurs.

- DNC operation

The control commands (such as GOTO and WHILE-DO) cannot be executed during DNC operation.
However, this restriction is removed when a program registered in program memory is called during DNC operation.

- Constant values that can be used in <expression>

+0.00000000001 to +999999999999
-999999999999 to -0.00000000001
The number of significant digits is 12 (decimal).
If this range is exceeded, an alarm PS0012 occurs.

16.13 INTERRUPTION TYPE CUSTOM MACRO

When a program is being executed, another program can be called by inputting an interrupt signal (UINT) from the machine.
This function is referred to as an interruption type custom macro function. Program an interrupt command in the following format:

Format

M96Pxxxxxxxx ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

Explanation

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.
(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.
(3) At regular intervals, information on current machining is read.
Listed above are examples like adaptive control applications of the interruption type custom macro function.

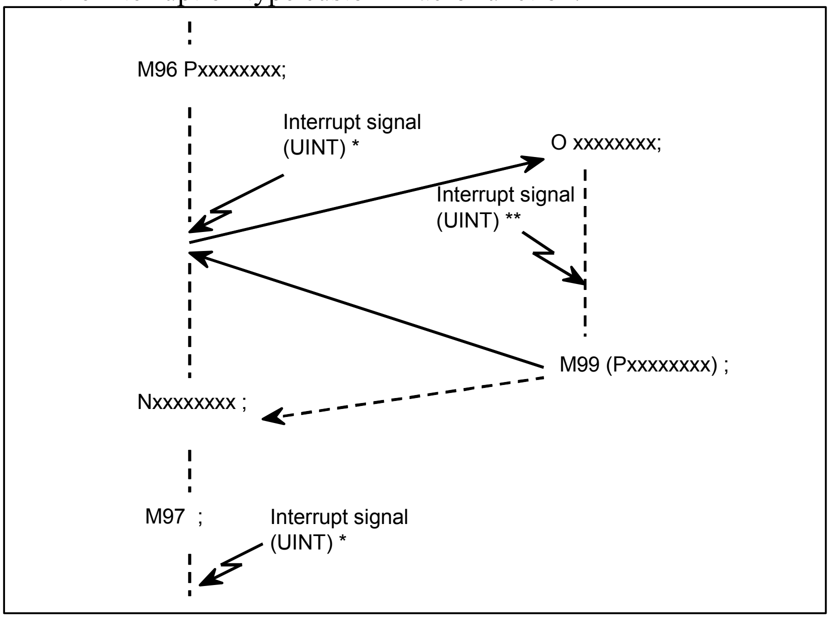


Fig 16.13 (a) Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx. When the interrupt signal (UINT, marked with an asterisk (*) in Fig 16.13 (a)) is input during execution of the interrupt program or after M97, it is ignored.

16.13.1 Specification Method

Explanation

- Interrupt conditions

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

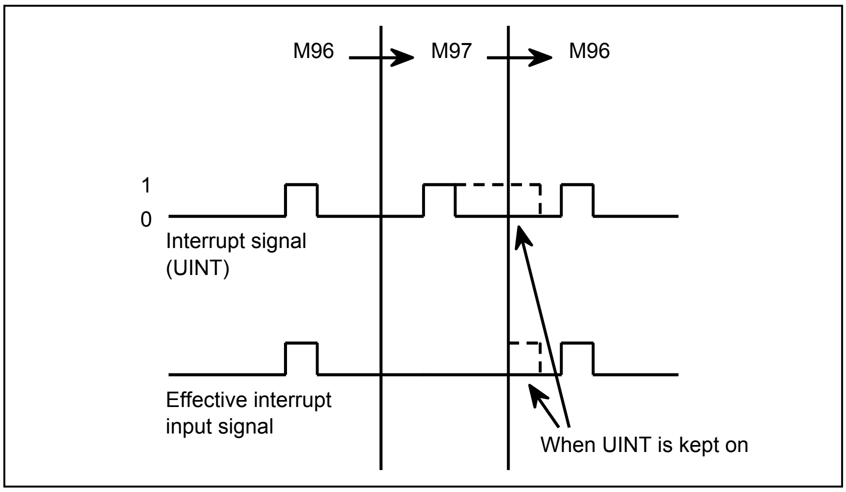
- When memory operation, DNC operation, or MDI operation is selected
- When STL (start lamp) is on
- When a custom macro interrupt is not currently being processed

A macro interrupt cannot be performed during manual operation.

- Specification

Generally, the custom macro interrupt function is used by specifying M96 to enable the interrupt signal (UINT) and M97 to disable the signal.

Once M96 is specified, a custom macro interrupt can be initiated by the input of the interrupt signal (UINT) until M97 is specified or the CNC is reset. After M97 is specified or the CNC is reset, no custom macro interrupts are initiated even when the interrupt signal (UINT) is input. The interrupt signal (UINT) is ignored until another M96 command is specified.



The interrupt signal (UINT) becomes valid after M96 is specified. Even when the signal is input in M97 mode, it is ignored. When the signal input in M97 mode is kept on until M96 is specified, a custom macro interrupt is initiated as soon as M96 is specified (only when the status-triggered scheme is employed); when the edge-triggered scheme is employed, the custom macro interrupt is not initiated even when M96 is specified.

NOTE

For the status-triggered and edge-triggered schemes, see Item "Custom macro interrupt signal (UINT)" of II-16.13.2.

16.13.2 Details of Functions

Explanation

- Subprogram-type interrupt and macro-type interrupt

There are two types of custom macro interrupts: Subprogram-type interrupts and macro-type interrupts. The interrupt type used is selected by MSB (bit 5 of parameter 6003).

(a) Subprogram-type interrupt: When MSB (bit 5 of parameter 6003) is set to 1
An interrupt program is called as a subprogram.
This means that the levels of local variables remain unchanged before and after the interrupt.
This interrupt is not included in the nesting level of subprogram calls.

(b) Macro-type interrupt: When MSB (bit 5 of parameter 6003) is set to 0
An interrupt program is called as a custom macro.
This means that the levels of local variables change before and after the interrupt.
The interrupt is not included in the nesting level of custom macro calls.
When a subprogram call or a custom macro call is performed within the interrupt program, this call is included in the nesting level of subprogram calls or custom macro calls.
Arguments cannot be passed from the current program even when the custom macro interrupt is a macro-type interrupt. The local variables immediately after interruption are all cleared to null.

- M codes for custom macro interrupt control

In general, custom macro interrupts are controlled by M96 and M97. However, these M codes, may already being used for other purposes (such as an M function or macro M code call) by some machine tool builders.
For this reason, MPR (bit 4 of parameter 6003) is provided to set M codes for custom macro interrupt control.
When specifying this parameter to use the custom macro interrupt control M codes set by parameters, set parameters 6033 and 6034 as follows:
Set the M code to enable custom macro interrupts in parameter 6033, and set the M code to disable custom macro interrupts in parameter 6034.
When specifying that parameter-set M codes are not used, M96 and M97 are used as the custom macro control M codes regardless of the settings of parameters 6033 and 6034.
The M codes used for custom macro interrupt control are processed internally (they are not output to external units). However, in terms of program compatibility, it is undesirable to use M codes other than M96 and M97 to control custom macro interrupts.

- Custom macro interrupts and NC statements

When performing a custom macro interrupt, the user may want to interrupt the NC statement being executed, or the user may not want to perform the interrupt until the execution of the current block is completed. MIN (bit 2 of parameter 6003)is used to select whether to perform interrupts even in the middle of a block or to wait until the end of the block. The type of interrupt performed even in the middle of a block is called type I and the type of interrupt performed at the end of the block is called type II.

> ⚠ **CAUTION**
> For interrupt type I, operation after control is returned differs depending on whether the interrupt program contains an NC statement.
> When the program number block contains EOB (;), it is assumed to contain an NC statement.
>
> | (Program containing an NC statement) | (Program containing no NC statement) |
> |---|---|
> | O0013; | O0013#101=#5041; |
> | #101=#5041; | #102=#5042; |
> | #102=#5042; | #103=#5043; |
> | #103=#5043; | M99; |
> | M99; | |

Type I (when an interrupt is performed even in the middle of a block)

(i) When the interrupt signal (UINT) is input, any movement or dwell being performed is stopped immediately and the interrupt program is executed.

(ii) If there are NC statements in the interrupt program, the command in the interrupted block is lost and the NC statement in the interrupt program is executed. When control is returned to the interrupted program, the program is restarted from the next block after the interrupted block.

(iii) If there are no NC statements in the interrupt program, control is returned to the interrupted program by M99, then the program is restarted from the command in the interrupted block.

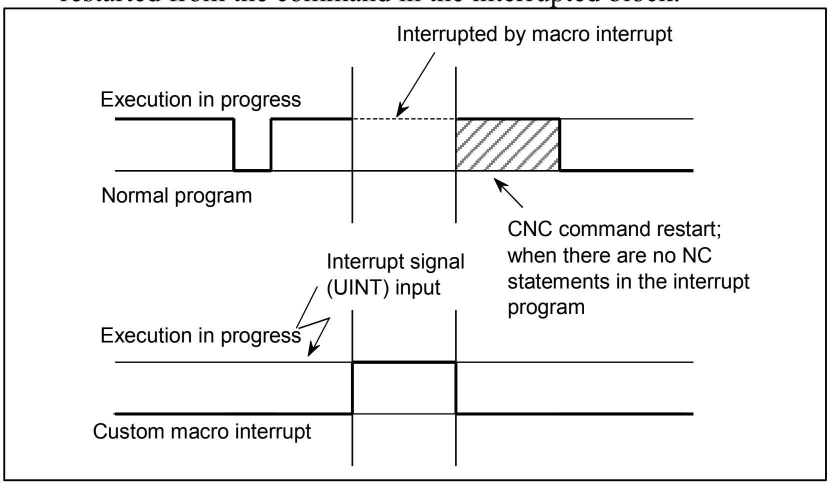


Fig. 16.13 (b) Custom macro interrupt and NC command (type I)

Type II (when an interrupt is performed at the end of the block)

(i) If the block being executed is not a block that consists of several cycle operations such as a drilling canned cycle and automatic reference position return (G28), an interrupt is performed as follows:
When an interrupt signal (UINT) is input, macro statements in the interrupt program are executed immediately unless an NC statement is encountered in the interrupt program. NC statements are not executed until the current block is completed.

(ii) If the block being executed consists of several cycle operations, an interrupt is performed as follows:
When the last movement in the cycle operations is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after all cycle operations are completed.

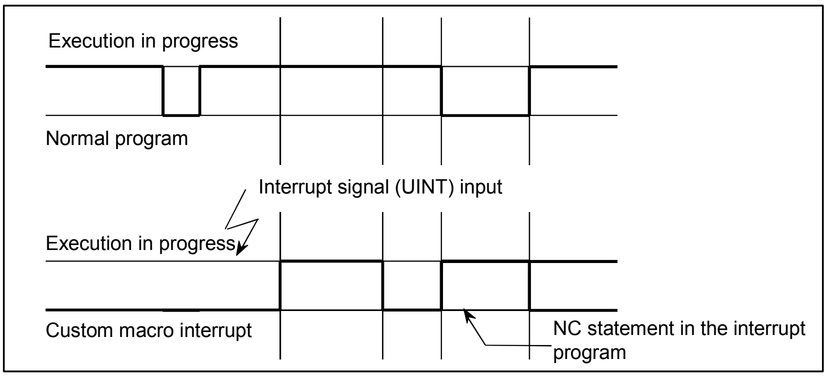


Fig. 16.13 (c) Custom macro interrupt and NC command (type II)

M

> **NOTE**
> During execution of a program for cycle operations, interrupt type II is performed regardless of whether bit 2 (MIN) of parameter No. 6003 is set to 0 or 1. Cycle operations are available for the following functions:
> <1> Automatic reference position return
> <2> Cutter or tool nose radius compensation (generating multiple blocks using the specified block such as when the tool moves around the outside of an acute angle)
> <3> Canned cycle
> <4> Automatic tool length measurement
> <5> Optional chamfering/corner R
> <6> Exponential interpolation
> <7> Normal direction control
> <8> Cutting point interpolation for cylindrical interpolation

T

> **NOTE**
> During execution of a program for cycle operations, interrupt type II is performed regardless of whether bit 2 (MIN) of parameter No. 6003 is set to 0 or 1. Cycle operations are available for the following functions:
> <1> Automatic reference position return
> <2> Cutter or tool nose radius compensation (generating multiple blocks using the specified block such as when the tool moves around the outside of an acute angle)
> <3> Canned cycle (No interruption type custom macro can be used during execution of a multiple repetitive canned turning cycle, however.)
> <4> Automatic tool length measurement
> <5> Chamfering/corner R
> <6> Exponential interpolation
> <7> Cutting point interpolation for cylindrical interpolation

- Conditions for enabling and disabling the custom macro interrupt signal

The interrupt signal becomes valid after execution starts of a block that contains M96 for enabling custom macro interrupts. The signal becomes invalid when execution starts of a block that contains M97.
While an interrupt program is being executed, the interrupt signal becomes invalid. The signal become valid when the execution of the block that immediately follows the interrupted block in the main program is started after control returns from the interrupt program. In type I, if the interrupt program consists of only macro statements, the interrupt signal becomes valid when execution of the interrupted block is started after control returns from the interrupt program.

- Custom macro interrupt signal (UINT)

There are two schemes for custom macro interrupt signal (UINT) input: The status-triggered scheme and edge-triggered scheme. When the status-triggered scheme is used, the signal is valid when it is on. When the edge triggered scheme is used, the signal becomes valid on the rising edge when it switches from off to on status.
One of the two schemes is selected with TSE (bit 3 of parameter 6003).

When the status-triggered scheme is selected by this parameter, a custom macro interrupt is generated if the interrupt signal (UINT) is on at the time the signal becomes valid. By keeping the interrupt signal (UINT) on, the interrupt program can be executed repeatedly.

When the edge-triggered scheme is selected, the interrupt signal (UINT) becomes valid only on its rising edge. Therefore, the interrupt program is executed only momentarily (in cases when the program consists of only macro statements). When the status-triggered scheme is inappropriate, or when a custom macro interrupt is to be performed just once for the entire program (in this case, the interrupt signal may be kept on), the edge-triggered scheme is useful.

Except for the specific applications mentioned above, use of either scheme results in the same effects. The time from signal input until a custom macro interrupt is executed does not vary between the two schemes.

In the example shown in Fig 16.13 (d), an interrupt is executed four times when the status-triggered scheme is used; when the edge-triggered scheme is used, the interrupt is executed just once.

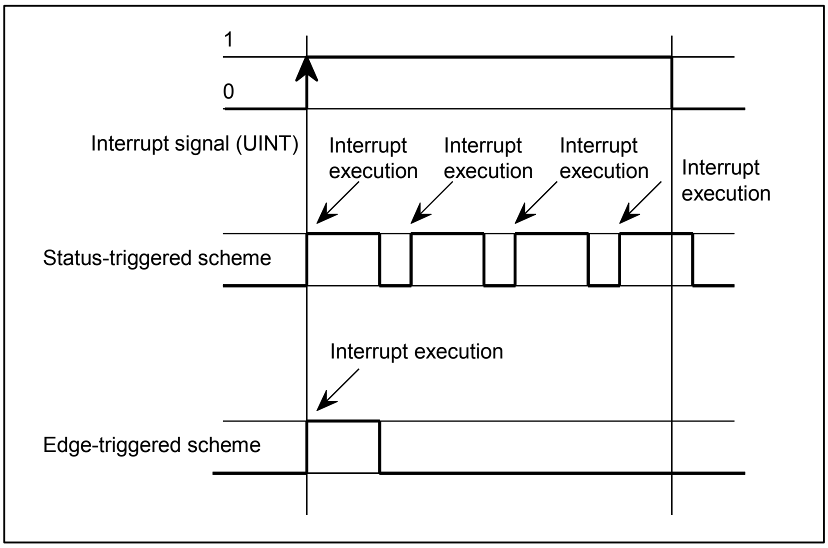


Fig. 16.13 (d) Custom macro interrupt signal

- Return from a custom macro interrupt

To return control from a custom macro interrupt to the interrupted program, specify M99. A sequence number in the interrupted program can also be specified using address P. If this is specified, the program is searched from the beginning for the specified sequence number. Control is returned to the first sequence number found.

NOTE

When an M99 block consists only of address O, N, P, L, or M, this block is regarded as belonging to the previous block in the program. Therefore, a single-block stop does not occur for this block. In terms of programming, the following <1> and <2> are basically the same. (The difference is whether Gxx is executed before M99 is recognized.)

```
<1>  Gxx Xxxx ;
     M99 ;
<2>  Gxx Xxxx M99 ;
```

- Custom macro interrupt and modal information

A custom macro interrupt is different from a normal program call. It is initiated by an interrupt signal (UINT) during program execution. In general, any modifications of modal information made by the interrupt program should not affect the interrupted program.
For this reason, even when modal information is modified by the interrupt program, the modal information before the interrupt is restored when control is returned to the interrupted program by M99.

When control is returned from the interrupt program to the interrupted program by M99 Pxxxxxxxx, however, modal information can again be controlled by the program. In this case, the new continuous information modified by the interrupt program is passed to the interrupted program.

In this case, take the following action as required:
<1> The interrupt program provides modal information to be used after control is returned to the interrupted program.
<2> After control is returned to the interrupted program, modal information is specified again as necessary.

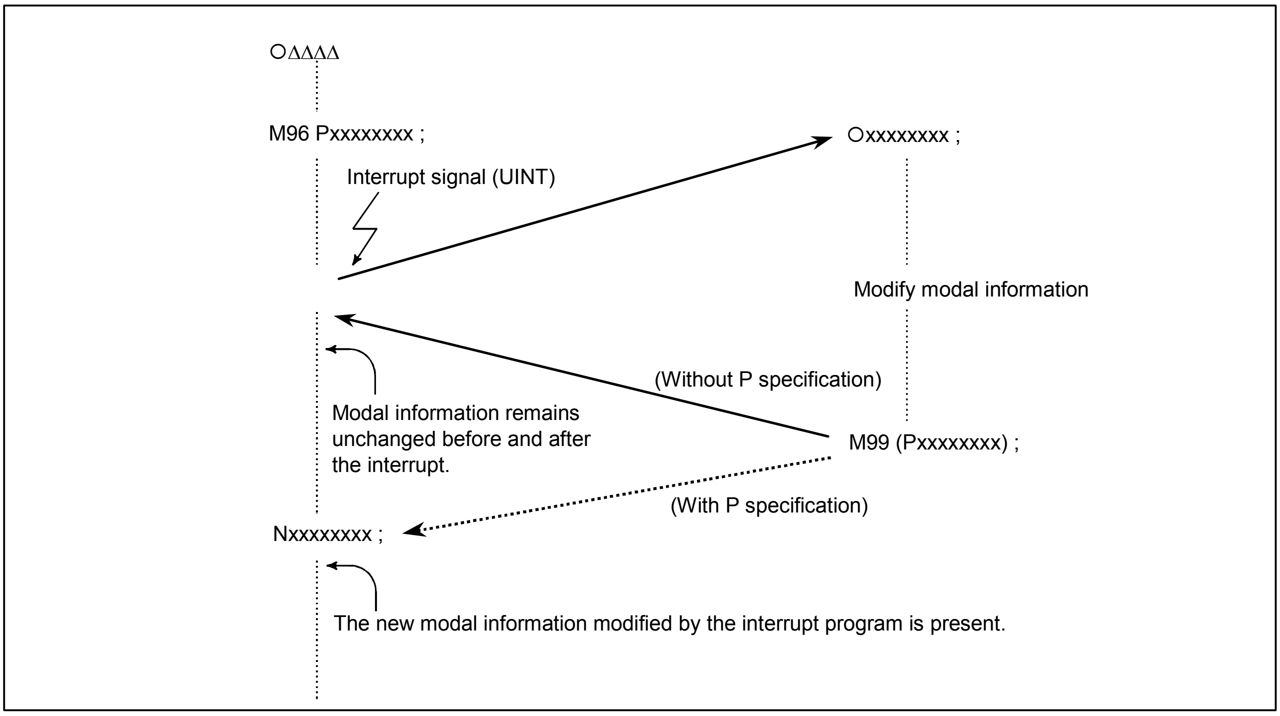


Fig. 16.13 (e) Custom macro interrupt and modal information

Modal information when control is returned by M99

The modal information present before the interrupt becomes valid. The new modal information modified by the interrupt program is made invalid.

Modal information when control is returned by M99 Pxxxxxxxx

The new modal information modified by the interrupt program remains valid even after control is returned.

Modal information which was valid in the interrupted block

The old modal information which was valid in the interrupted block can be read using custom macro system variables #4401 to #4530.

M

System variable	Modal information which was valid when a custom macro interrupt was generated
#4401	G code (group 01)
:	:
#4421	G code (group 21)
#4502	B code
#4507	D code
#4508	E code
#4509	F code
#4511	H code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code
#4530	Additional workpiece coordinate system number

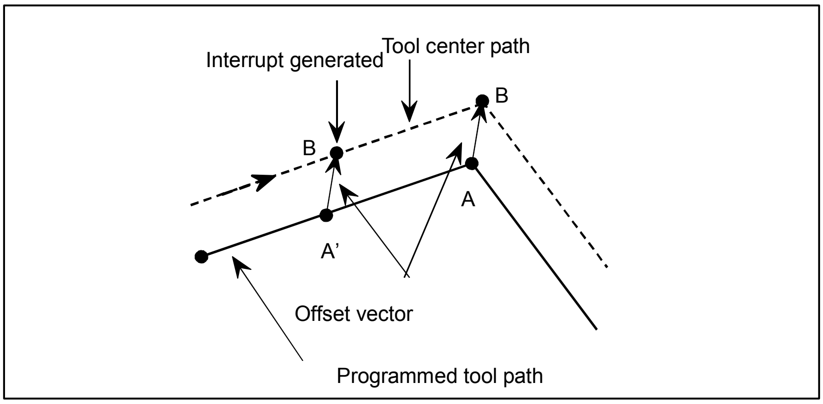
T

System variable	Modal information which was valid when a custom macro interrupt was generated
#4401	G code (group 01)
:	:
#4421	G code (group 21)
#4508	E code
#4509	F code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code
#4530	Additional workpiece coordinate system number

- System variables (position information values) for the interrupt program

Position information can be read as follows.

Macro variable	Condition	Position information value
#5001 or above	Until the first NC statement appears	Coordinates of point A
	After an NC statement with no move command appears	Coordinates of point A'
	After an NC statement with a move command appears	Coordinates of the end point of the move command
#5021 or above		Machine coordinates of point B'
#5041 or above		Workpiece coordinates of point B'



- Custom macro interrupt and custom macro modal call

When the interrupt signal (UINT) is input and an interrupt program is called, the custom macro modal call is canceled (G67). However, when G66 is specified in the interrupt program, the custom macro modal call becomes valid. When control is returned from the interrupt program by M99, the modal call is restored to the state it was in before the interrupt was generated. When control is returned by M99 Pxxxxxxxx;, the modal call in the interrupt program remains valid.

- Custom macro interrupt and program restart

In program restart, when the interrupt signal (UINT) is input during dry run recovery after a search, the interrupt program is called after restart of all axes is completed.
That is, interrupt type II is assumed regardless of the parameter setting.

M

> **NOTE**
> 1 Alarm PS1101 occurs in the following cases:
> <1> An interrupt is generated in the programmable mirror image (G51.1) mode and another G51.1 is specified in the interrupt program.
> <2> An interrupt is generated in the coordinate system rotation (G68) mode and another G68 is specified in the interrupt program.
> <3> An interrupt is generated in the scaling (G51) mode and another G51 is specified in the interrupt program.
> 2 In program restart, do not input the interrupt signal (UINT) during dry run recovery after a search.

T

> **NOTE**
> 1 Alarm PS1101 occurs in the following cases:
> <1> An interrupt is generated in the programmable mirror image (G51.1) mode and another G51.1 is specified in the interrupt program.
> <2> An interrupt is generated in the coordinate system rotation (G68.1) mode and another G68.1 is specified in the interrupt program.
> <3> An interrupt is generated in the scaling (G51) mode and another G51 is specified in the interrupt program.
> 2 No interruption type custom macro can be used during execution of a multiple repetitive canned turning cycle.
> 3 In program restart, do not input the interrupt signal (UINT) during dry run recovery after a search.

17 REAL-TIME CUSTOM MACRO

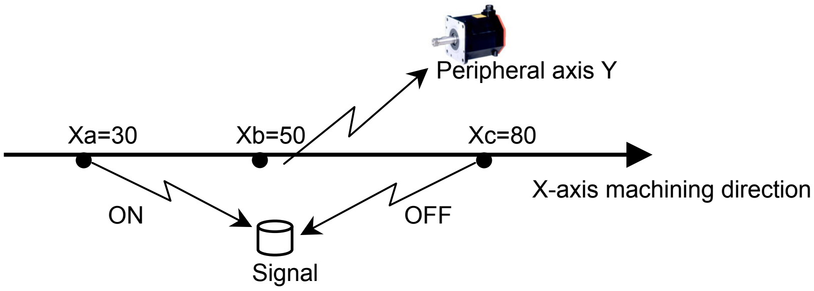
Overview

Used with an NC program, the real time custom macro function controls peripheral axes and signals.

If a macro statement is used together with an NC statement, a program using the conventional custom macro function executes the macro statement immediately when the macro statement is read. So, the macro statement cannot be executed independently of the NC statement.
On the other hand, the real time custom macro function enables the following control operations when a real time macro command (RTM command) is coded in an NC program.

- A real time macro command starts operation in synchronism with the NC statement and is executed independently. During NC program execution, a real time macro command can be executed at the same time.
- PMC interface signals can be read and written (with a restriction). In an NC program, a motion using a signal as a trigger can be coded.
- Variables dedicated to a real time macro command can be read and written.
- A real time macro command can exercise axis control. (The option is required.)
- Multiple real time macro commands can be executed at the same time. Multiple real time custom macro statements can be coded in an NC program and can be controlled independently of each other.

By using a real time custom macro, control on signals and peripheral axes can be programmed.
The example below turns on and off a signal and performs operation on a peripheral axis when machining points are passed.

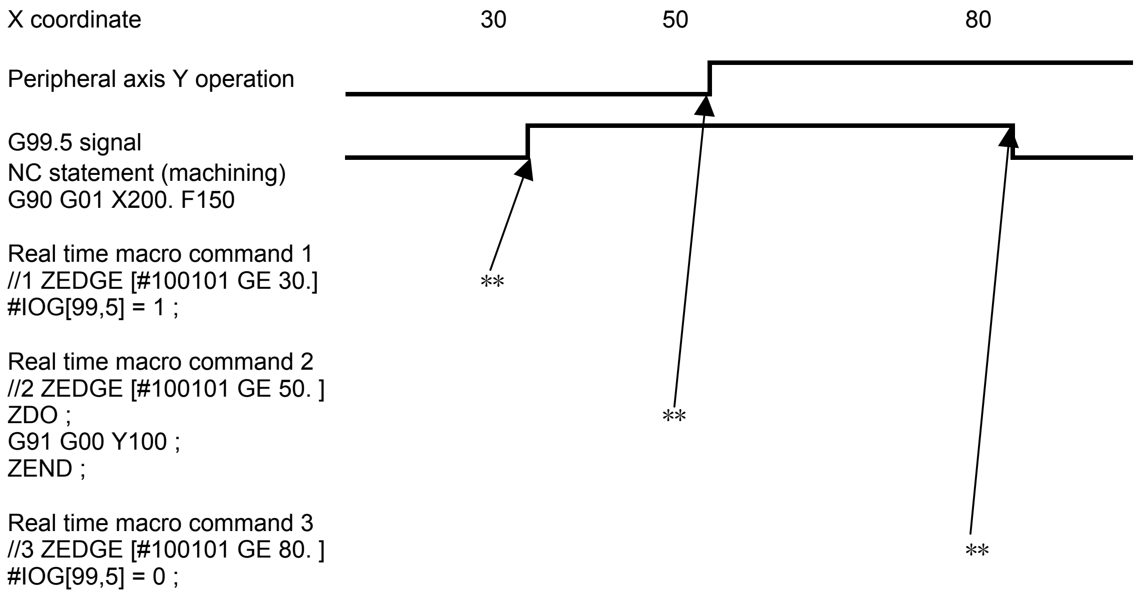


During movement for machining along the X-axis in the figure above:
<1> When point Xa is passed, the G99.5 signal is set to 1.
→ Real time macro command 1
<2> When point Xb is passed, positioning is started on peripheral axis Y. → Real time macro command 2
<3> When point Xc is passed, the G99.5 signal is set to 0.
→ Real time macro command 3

The operation above is programmed using real time macro commands.

```
Program
O0001 ;
G92 X0 ;
//1 ZEDGE [#100101 GE 30. ] #IOG[99,5] = 1 ;
//2 ZEDGE [#100101 GE 50.] ZDO ;
G91 G00 Y100 ;
ZEND ;
//3 ZEDGE [#100101GE 80. ] #IOG[99,5] = 0 ;
G90 G01 X200. F150 ;
M30 ;
```

The timing is as follows. ('**' represents the satisfaction of the condition.)



Explanation

To use the real time custom macro function, a real time macro command (RTM command) is coded in an NC program.

- Real time macro command (RTM command)

The real time macro command (RTM command) is a macro command that starts execution in phase with an NC statement in the program. After the execution of a real time macro command starts, the RTM command operates independently of the NC statement.
An RTM command is a statement dedicated to the real time custom macro function.
An RTM command consists of a set of one or more real time macro statements (RTM statements).

- Real time macro statement (RTM statement)

The real time macro statement (RTM statement) is a statement included in an RTM command.
One or more RTM statements make up an RTM command.

An RTM statement consists of a macro command and axis control command dedicated to the real time custom macro function.

The axis control command of an RTM statement is an RTM statement including an address. This command is used to exercise axis control.
Example

```
// ZDO ;
G90 G00 X100 ;
ZEND ;
```

(ZDO and ZEND are reserved words required for the axis control command of an RTM statement, and are detailed later.)

The macro command of an RTM statement is a macro statement used with an RTM statement for arithmetic/logical operation and signal control. (In the example below, #RV[1] and #RV[2] are variables dedicated to the real time macro command.)
Example

```
// #IOG[124, 5] = 1 ;
// #RV[1] = #RV[2] * 10 ;
```

Format

The real time macro command format is shown below.
The RTM command is a command with two slashes (//) prefixed at the start of a block.

//n <real-time-macro-statement> **or** **//n ZDO ;** **<real-time-macro-statement>** **:** **ZEND ;**
N: Modal ID (1 to 10) (Omissible) When a proper number is coded in n, a modal real time macro command is specified. When n is omitted, a one-shot real time macro command is specified.

ZDO to ZEND are detailed later.

17.1 TYPES OF REAL TIME MACRO COMMANDS

17.1.1 Modal Real Time Macro Command / One-shot Real Time Macro Command

Explanation

A command with '//' followed by an RTM statement is referred to as a one-shot real time macro command (one-shot RTM command).
Example:
// #RV[1] = 30 ;

On the other hand, a command with '//' followed by number n (1 to 10) then an RTM statement is referred to as a modal real time macro command (modal RTM command).
Example:
//3 #RV[1] = 30 ;

A one-shot RTM command starts when the execution of the first following NC command starts. A started one-shot RTM command is valid until the end of the NC command.
A modal RTM command starts when the execution of the first following NC command starts, as in the case of a one-shot RTM command.
Unlike a one-shot RTM command, however, a started modal RTM command is valid until the automatic operation is completed.

- Start of a real time macro command

An RTM command starts when the execution of the first following NC command starts.

Example:
When NC command (1) starts execution in the program below, macro commands (2) and (4) are executed in succession without waiting for the end of (1).
On the other hand, RTM command (3)starts execution when NC command (5) starts execution after the end of NC command (1).

```
O0001 ;
G90 G00 X30. ;      (1) NC command
#100=0              (2) Macro command
// #RV[0]=1 ;       (3) RTM command
#102=2;             (4) Macro command
G90 G00 X100. ;     (5) NC command
M30 ;
```

- End of a real time macro command

When one of the following conditions is satisfied, the RTM command is terminated.

Termination conditions common to one-shot RTM and modal RTM commands

- When RTM command processing is completed
- When a reset occurs

Termination condition specific to a one-shot RTM command

- When the execution of the NC command that started at the same time has ended

However, if the RTM statement being executed is an axis control command, the command is terminated when the execution of the block is completed.
If the execution of the Y10. block is completed before the X100. block in the command below, and the execution of the Y20. block starts, for example, the X100. command of the RTM statement is executed until the end.

```
// ZDO ;
X100 ;
ZEND ;
Y10. ;
Y20. ;
```

NOTE

1 No one-shot RTM command can be specified using any of the commands indicated below as a trigger. When using any of these commands as a trigger, use a modal RTM command.
   - Command related to reference position return
   - Command related to AI contour control
   - Command related to canned cycles (rigid tapping, drilling cycle, etc.)
   - Command related to cutter compensation
   - Command related to tool length compensation
   - Command related to automatic tool length measurement
   - Command related to coordinate system rotation Command related to scaling
   - Command related to programmable mirror image

2 If an RTM command is specified using, as a trigger, a block such as a block specifying NURBS interpolation or a T series multiple repetitive canned cycle that does not necessarily pass the start point or end point of the command, operation can start or end at a point other than the start point and end point. Do not use such a block as a trigger.

> **NOTE**
> 3 Do not restart a program that includes an RTM command.
> 4 When an NC statement used as a trigger for an RTM command represents an auxiliary function, execution continues even if the FIN signal is awaited.
> If the following program is executed, for example, the count-up operation of #RV[0] continues until the FIN signal of M55 is returned:
> ```
> O0001 ;
> // ZWHILE [1] #RV[0] = #RV[0]+1 ;
> M55 ;
> G91 X200. ;
>   :
> ```
> 5 If M02 follows an RTM command, execution continues until a reset occurs, even when the program itself is terminated.
> If the following program is executed, for example, the count-up operation of #RV[0] continues until a reset occurs:
> ```
> O0001 ;
> // ZWHILE [1] #RV[0] = #RV[0]+1 ;
> M02 ;
> ```

- Priority of commands

If a modal RTM command and one-shot RTM command are specified at the same time, the modal RTM command is executed first.
If multiple modal RTM commands are specified at the same time, the commands are executed in ascending order of ID values.
No priority is applicable to one-shot RTM commands.
The order of execution is as follows:

```
Modal RTM command with ID value being 1      High
Modal RTM command with ID value being 2       ▲
      :                                       |
      :                                       |
Modal RTM command with ID value being 10      |
One-shot RTM command                          |
      :                                       |
One-shot RTM command                          ▼
                                             Low
```

Example 1)

Priority of modal RTM commands

```
O0001 ;
//1 #RV[0]=1 ;
//3 #RV[0]=3 ;
//2 #RV[0]=2 ;
M02 ;
```

When the program above is executed, the RTM commands are executed in the following order:
#RV[0]=1
#RV[0]=2
#RV[0]=3
So, the value of #RV[0] is 3.

Example 2)
Priority of modal RTM commands and a one-shot RTM command
O0001 ;
//3 #RV[0]=3 ;
//1 #RV[0]=1 ;
// #RV[0]=10 ;
//5 #RV[0]=5 ;
M02 ;

When the program above is executed, the RTM commands are executed in the following order:
//1 #RV[0]=1
//3 #RV[0]=3
//5 #RV[0]=5
// #RV[0]=10
So, the value of #RV[0] is 10.

Example 3)
Priority of one-shot RTM commands
When the following program is executed, the value of #RV[0] is undefined, that is, 1, 2, or 3.
O0001 ;
//#RV[0]=1 ;
//#RV[0]=2 ;
//#RV[0]=3 ;
G04P10 ;
M30 ;

Even if a priority is applicable, the order of execution or the order of termination can change when an RTM statement includes a control code, ZWHILE or ZEDGE, or axis control command.

Example 4)
The RTM command priority of #RV[0]=1 in a modal command with its ID value being 1 is higher than the priority of #RV[1]=1 in a modal command with its ID value being 2. However, #RV[0]=1 is executed after the end of the execution of the block specifying the axis control command G91 G00 X10., so that #RV[1]=1 is actually executed earlier than #RV[0]=1.
O0001 ;
//1 ZDO ;
G91 G00 X10. ;
#RV[0]=1 ;

```
ZEND ;
//2 #RV[1]=1 ;
G04 P10 ;
M30 ;
```

Example 5)

In the RTM command priority, ZEDGE in a modal command with its ID value being 1 is always a false control code (detailed later). The RTM command priority of #RV[0]=1 in a modal command with its ID value being 1 is higher than the priority of #RV[1]=1 and #RV[2]=1 in a modal command with their ID values being 2. However, #RV[0]=1 is executed after the condition of ZEDGE becomes true (that is, at the second time or later), so that #RV[1]=1 and #RV[2]=1 are executed earlier than #RV[0]=1.

```
O0001 ;
//1 ZEDGE [ #IOG[234.0] EQ 1 ] #RV[0]=1 ;
//2 ZDO ;
#RV[1]=1 ;
#RV[2]=1 ;
ZEND ;
G04 P10 ;
M30 ;
```

- Number of real time macro commands

A program can have multiple RTM commands coded.
Up to six one-shot RTM commands can be specified.
If one-shot RTM commands more than the maximum allowable number are specified, an alarm is issued.

Up to ten modal RTM commands can be specified. When specifying modal RTM commands, ensure that there is no duplicate ID.

If there is a duplicate ID, or an incorrect ID is specified, an alarm is issued.

In all paths, up to 16 RTM commands can be executed simultaneously.
When an axis control command is included, up to four commands can be executed simultaneously.

> **NOTE**
> 1 In a block specifying an RTM statement, no NC command can be coded.
> 2 If the maximum specifiable number of commands or the maximum number of simultaneously executable commands is exceeded, a P/S alarm is issued.

NOTE

3 If an NC statement to be used to trigger an RTM command is specified in a block (e.g., small block) that ends in a very short time, an RTM statement programmed to start at a different timing may be executed simultaneously. If the following is specified, for example, #RV[0]=1 and #RV[1]=2 may be executed simultaneously:

```
// #RV[0]=1 ;
G91 G01 X0.002 ; F5000
// #RV[1]=2 ;
X0.001 ;
```

4 When a function for reading multiple blocks in advance is used, the same ID must not be coded in the scope of blocks read in advance. In the following program, for example, while //1 #RV[0]=#100101 is being executed during AI contour control, //1 #RV[1]=#100101 and //1 #RV[2]=#100101 are also read in advance. So, an alarm may be issued.

```
//1 #RV[0]=#100101 ;
X#100 ;
//1 #RV[1]=#100101 ;
X#100 ;
//1 #RV[2]=#100101 ;
X#100 ;
```

5 When a function for reading multiple blocks in advance is used, up to three blocks among the blocks read in advance can trigger an RTM command.
For example, if the blocks up to the block of (2) are read in advance during execution of (1) in the program below, up to three blocks can trigger an RTM command. In the program below, the number of NC blocks that trigger an RTM command exceeds 3, so that the RTM command of (a) must not be coded.

```
X30. Y50. ;    .....................................................(1)
// Z-30. ;
// #RV[0]=#RV[0]+1 ;
X3. Y16. ; → NC block 1 triggering an RTM command
X-23. Y4. ;
// #RV[1]=#RV[1]+1 ;
//2 Z30. ;
X-2. Y9. ; →  NC block 2 triggering an RTM command
X17. Y5. ;
// #RV[2]=#RV[2]+1 ;
X-2. Y9. ; →  NC block 3 triggering an RTM command
// #RV[3]=#RV[3]+1 ; ..........................................(a)
X-12. Y-3. ;
X-100. Y200. ; ...................................................(2)
```

- Reserved words

The following reserved words are used with real time custom macros:

- Reserved words dedicated to real time custom macros
  ZDO, ZEND, ZONCE, ZWHILE, ZEDGE

- Reserved words shared with custom macros
  AND, OR, XOR, MOD, EQ, NE, GT, LT, GE, LE, SIN, COS, TAN, ASIN, ACOS, ATAN, ATN, SQRT, SQR, ABS, BIN, BCD, ROUND, RND, FIX, FUP, LN, EXP, POW

Be sure to fully spell out the reserved words for real time custom macros. For example, 'ZONCE' must not be coded as 'ZON' or 'ZONC'.

17.2 VARIABLES

Overview

With real time custom macros, the following variables can be handled:

- System variables dedicated to real time custom macros
- Variables (RTM variables) dedicated to real time custom macros
- System variables for some custom macros

The RTM variables mean the variables dedicated to real time custom macros.

List of usable variables

		Real time custom macro	Custom macro
Real time custom macro variables	System variables	Usable	Unusable
	RTM variables	Usable	Unusable
Custom macro variables	System variables	Partially usable	Usable
	Common variables	Unusable	Usable
	Local variables	Unusable	Usable

The variables (system variables and RTM variables) dedicated to real time custom macros are the variables specific to the real time custom macro function. Those variables cannot be used with the custom macro function.

17.2.1 Variables Dedicated To Real Time Custom Macros

These variables are dedicated to real time custom macros. The variables are classified as system variables and RTM variables.

17.2.1.1 System variables

System variables dedicated to real time custom macros

Format

#IOp [m, n] Bit-by-bit read/write
#IOpB [m] Byte-by-byte read/write
p : Signal type (X, G, F, Y)
m : Signal byte address
n : (Used for bit-by-bit read/write only) Signal address bit number (0 to 7)

PMC interface signals are read and written. Bit-by-bit and byte-by-byte read/write operations are possible.
The following signals can be used:

Variable name	Signal type	Read	Write
#IOX #IOXB	X	Possible	Impossible
#IOG #IOGB	G	Impossible	Possible
#IOF #IOFB	F	Possible	Impossible
#IOY #IOYB	Y	Impossible	Possible

For the valid signal address range, see the specifications of the PMC as well.

When writing to a signal, make the variable unprotected on the PMC signal protection screen (described later) beforehand.

Specify an address by using m and n.
Example:

```
#IOF[1, 3]      F1.3 bit type
#IOG[1, 5]      G1.5 bit type
#IOFB[32]       F32 byte type
#IOGB[12]       G12 byte type
```

Read/write operations are performed in the same as for an ordinary macro statement.
Example:

```
#RV[0]=#IOFB[32]    Assigns F32 to #RV[0].
#IOG[99.3] = 1             Sets G99.3 to 1.
```

If a signal at a nonexistent address is specified, a P/S alarm is issued.

> ⚠ **CAUTION**
> 1 Controls handling other signals, such as a ladder or macro executor, must not write to a signal address being written to by an RTM statement. Ensure that a single control writes to the same byte signal address.
> For example, when the G000.0 signal is written to by an RTM statement, do not write to the G000.7 signal from a ladder.
> 2 Ensure that the same F signal is not read from by an RTM statement and written to by the NC simultaneously.
> 3 Ensure that the same G signal is not written to by an RTM statement and read from by the NC simultaneously.

> **NOTE**
> The system variables described above are not supported for a multi-path PMC.

- PMC signal protection

Whether to enable a write to a signal handled by an RTM statement can be set. This function protects against a malfunction due to incorrect coding.

On the PMC signal protection screen, set whether to enable a write to a signal.
If an RTM statement makes an attempt to write to a signal write-protected on the PMC signal protection screen, a P/S alarm is issued at the time of execution.

Set whether to enable a write to each address of Y and G on a byte-by-byte basis.

For the unwritable signals (X, F), the screen is not displayed.

- Input/output

A value set for PMC signal protection can be input/output.

- Input/output format

After punching PMC signal protection, one file(DIDOENBL.TXT) is created.
Please execute input/output operation in EDIT mode.
The output format is as follows:
L Specification method
0: Byte specification
Q Alphabetic signal address
0: G, 2: Y
K Not used

R Address number for byte specification
P Protection value for byte specification
0: Not writable
1: Writable

Example of output

```
%
L0Q2R0000P0
L0Q2R0001P1          Y0 to Y127 protect
:                    information
L0Q2R0127P1
:
L0Q0R0000P1                                       Byte
L0Q0R0001P1          G0 to G767 protect           specification
:                    information
L0Q0R0767P0
L0Q0R1000P1
:                    G1000 to G1767 protect
L0Q0R1767P0          information
:
L1Q-2K0R0000P0000
:
L1Q-2K0R0000P0000
M02
%
```

17.2.1.2 Real time macro variables (RTM variables)

The real time macro variables (RTM variables) are variables dedicated to real time custom macros.
The RTM variables are classified as volatile real time macro variables (volatile RTM variables) and nonvolatile real time macro variables (nonvolatile RTM variables).

The data of a nonvolatile RTM variable is preserved even when the power is turned off.
The data of a volatile RTM variable is cleared to 0 when the power is turned off.

Format

#RV [m] Volatile RTM variable
m: Volatile RTM variable number (0 to 99)
#RVS [n] Nonvolatile RTM variable
n: Nonvolatile RTM variable number (0 to 31)

> **NOTE**
> 1 RTM variables can be used with an RTM statement only. RTM variables cannot be used with an NC statement and macro statement.
> 2 No RTM variable assumes a "null" value.
> 3 Volatile RTM variables are cleared to 0 by a reset. On the other hand, nonvolatile RTM variables are not cleared to 0 by a reset.

Explanation

- Input/output

RTM variables can be input/output in a specified format.
Both nonvolatile RTM variables and volatile RTM variables can be input/output.

- Input/output format

After punching RTM variables, one file(RTMMACRO.TXT) is created.
The output format is described below.
The value of an RTM variable is output in hexadecimal as a double-precision floating-point data bit image.
When a nonvolatile RTM variable is output, G10L87 is followed by an RTM variable number and a variable value.
When a volatile RTM variable is output, G10L88 is followed by an RTM variable number and a variable value.

Example of output

```
%
G10L87P0(3FE0000000000000)  ⎫
G10L87P1(4000000000000000)  ⎪
:                           ⎬ Nonvolatile RTM variable
G10L87P30(4010000000000000) ⎪
G10L87P31(4014000000000000) ⎭

G10L88P0(4008000000000000)  ⎫
G10L88P1(3FD9999999800000)  ⎪
:                           ⎬ Volatile RTM variable
G10L88P98(3FF0000000000000) ⎪
G10L88P99(4010000000000000) ⎭
M02
%
```

Execute input/output operation in EDIT mode.

17.2.2 Custom Macro Variables

With real time custom macros, a part of the custom macro variables (part of the system variables) can be handled.

17.2.2.1 System variables

With real time custom macros, position-related information among the system variables of the custom macros can be handled.

- Position information #100001 to #100182 (Attribute: Read only)

Block end position #100001 to #100032
Current position (machine coordinate system) #100051 to #100082
Current position (workpiece coordinate system) #100101 to #100132
Skip position #100151 to #100182

By readying the values of system variables #100001 to #100182, the end position of the previous block, the current positions (machine coordinate system and workpiece coordinate system), and skip signal position can be found.

Variable No.	Position information	Coordinate system
#100001 #100002 : #100032	Block end position on 1st axis Block end position on 2nd axis : Block end position on 32nd axis	Workpiece coordinate system
#100051 #100052 : #100082	Current position on 1st axis Current position on 2nd axis : Current position on 32nd axis	Machine coordinate system
#100101 #100102 : #100132	Current position on 1st axis Current position on 2nd axis : Current position on 32nd axis	Workpiece coordinate system
#100151 #100152 : #100182	Skip position on 1st axis Skip position on 2nd axis : Skip position on 32nd axis	Workpiece coordinate system

NOTE

1 The value of a variable with a number greater than the number of controlled axes is undefined.
2 The end position (ABSIO) of a skip (G31) block is the skip signal ON position if the skip signal is turned on. If the skip signal is not turned on, the block end point is undefined.
3 As block end position information #100001 to #100032, the end positions read in advance are obtained instead of the end positions of the block currently being executed.

- Servo positional deviation #100251 to #100282 (Attribute: Read only)

By reading the values of system variables #100251 to #100282, the servo positional deviation on each axis can be found.

Variable No.	Position information
#100251	Servo positional deviation on 1st axis
#100252	Servo positional deviation on 2nd axis
:	:
#100282	Servo positional deviation on 32nd axis

> **NOTE**
> The value of a variable with a number greater than the number of controlled axes is undefined.

- Remaining travel distance #100801 to #100832 (Attribute: Read only)

By reading the values of system variables #100801 to #100832, the remaining travel distance on each axis can be read.

Variable No.	Position information
#100801	Travel distance on 1st axis
#100802	Travel distance on 2nd axis
:	:
#100832	Travel distance on 32nd axis

> **NOTE**
> 1 The value of a variable with a number greater than the number of controlled axes is undefined.
> 2 System variables not described here are unusable.
> 3 The name of a system variable cannot be specified.
> Do not specify a command as indicated below.
> //1 #RV[0]=[#_ABSOT[1]] ;

17.2.2.2 Local variables

The local variables(#1 to #33) cannot be used.

17.3 ARITHMETIC AND LOGICAL OPERATION

With the real time custom macros, the following arithmetic and logical operations can be specified:

Table 1.3 Arithmetic and logical operation

Type of operation	Operation	Description
(1) Definition, substitution	#i=#j	Definition or substitution of a variable
(2) Additive operation	#i=#j+#k #i=#j-#k #i=#j OR #k #i=#j XOR #k	Addition Subtraction Logical OR (Bit by bit for 32 bits) Exclusive OR (Bit by bit for 32 bits)
(3) Multiplicative operation	#i=#j*#k #i=#j/#k #i=#j AND #k #i=#j MOD #k	Multiplication Division Logical AND (Bit by bit for 32 bits) Remainder (#j and #k are rounded to an integer to find a remainder. When #j is negative, #i is also negative.)
(4) Function	#i=SIN[#j] #i=COS[#j] #i=TAN[#j] #i=ASIN[#j] #i=ACOS[#j] #i=ATAN[#j] #i=ATAN[#j]/[#k] #i=ATAN[#j,#k] #i=SQRT[#j] #i=ABS[#j] #i=BIN[#j] #i=BCD[#j] #i=ROUND[#j] #i=FIX[#j] #i=FUP[#j] #i=LN[#j] #i=EXP[#j] #i=POW[#j,#k]	Sine (in deg) Cosine (in deg) Tangent (in deg) Arcsine Arccosine Arctangent (1 argument). ATN is acceptable. Arctangent (2 arguments). ATN is acceptable. Ditto Square root. SQR is acceptable. Absolute value Binary conversion from BCD BCD conversion from binary Rounding. RND is acceptable. Discarding fractional digits Rounding fractional digits upward to an integer Natural logarithm Exponent with e (2.718……) used as the base Power (#j to #k power)

- Constant specifiable in <expression>

+0.00000000001 to +999999999999
-999999999999 to -0.00000000001
Up to 12 decimal digits can be specified.
If the maximum allowable number of digits is exceeded, P/S0012 alarm is issued.

NOTE

1. The ADP function is not available.
2. With an RTM statement, the external output commands (BPRNT, DPRNT, POPEN, and PCLOS) are unavailable.
3. The FS16i compatibility specifications are not applicable. Bit 0 (F16) of parameter No. 6008 = 1 (with the operation result precision based on the FS16i compatibility specifications) is invalid.
4. Bit 0 (NAT) of parameter No. 6004 is valid in an RTM command.
5. The setting of bit 5 (SBM) of parameter No. 6000 and bit 7 (SBV) of parameter No. 6000 is invalid. For the single block specifications, see the separate item.

17.4 CONTROL ON REAL TIME MACRO COMMANDS

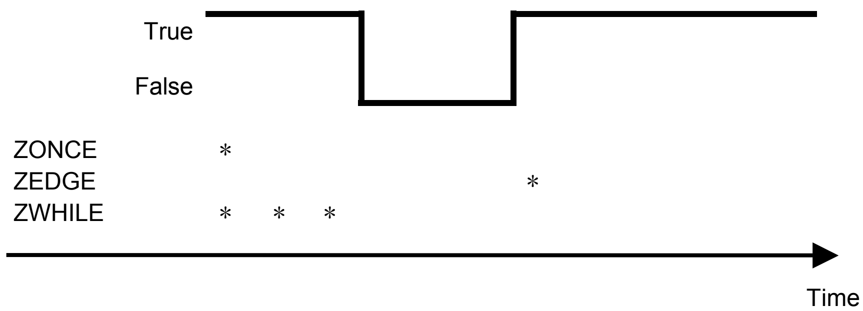
Explanation

By using a reserved word for controlling statements in an RTM command, the flow of the RTM command can be changed or multiple statements can be controlled as a set of statements. Four reserved words are used to control an RTM command as indicated below.

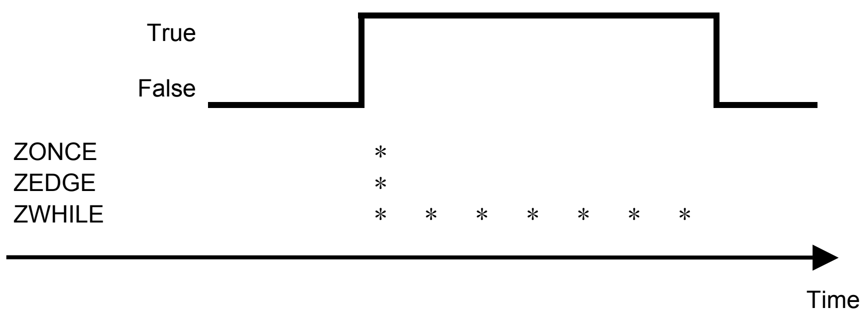
Reserved word name	Syntax	Meaning
ZONCE	// ZONCE A B	If A is true, B is executed.
ZEDGE	// ZEDGE A B	If A becomes true, B is executed.
ZWHILE	// ZWHILE A B	While A is true, B is executed repeatedly.
ZDO...ZEND	// ZDO B1 B2 B3 ZEND	(Multiple statements) B1, B2, and B3 are sequentially executed.

The timing chart of an RTM command using these reserved words is indicated below. (Multi-statement control ZDO...ZEND is excluded.)
When the condition of a each reserved word is True, it shows '*'.

When condition A makes transitions from True to False to True



When condition A makes transitions from False to True to False



17.4.1 Conditional Branch (ZONCE Statement)

After ZONCE, <conditional-expression> and <real-time-macro-statement> are coded.

- //(n) ZONCE [<conditional-expression>] <real-time-macro-statement>

If <conditional-expression> is true, <real-time-macro-statement> is executed. If <conditional-expression> is false, <real-time-macro-statement> is not executed, but control exits from the ZONCE command to terminate the execution. <conditional-expression> conforms to the custom macro specifications.

// ZONCE A B (If A is true, B is executed.)
If A is true, the command is terminated after executing B.
If A is false, the command is terminated without executing B.

If the workpiece coordinate on the first axis is greater than 30, the Y1.0 signal is output.
// ZONCE [#100101 GT 30.] #IOY[1,0] = 1 ;

If the workpiece coordinate on the first axis is greater than 30 and #RV[0] matches #RV[3], the Y1.0 signal is output.
// ZONCE[[#100101 GT 30.] AND [#RV[0] EQ #RV[3]]] #IOY[1,0]=1 ;

In <real-time-macro-statement>, multiple RTM statements can be coded.
In this case, code the following by using ZDO...ZEND of multi-statement structure:
// ZONCE [<conditional-expression>] ZDO ;
<real-time-macro-statement-1> ;
<real-time-macro-statement-2> ;
:
ZEND ;

If the workpiece coordinate on the second axis is equal to or less than 10, the rapid traverse override value is changed.
// ZONCE [#100102 LE 10.] ZDO ; **#IOG[14,0]=0 ;** **#IOG[14,1]=1 ;** **ZEND ;**

However, if <conditional-expression-1> specifies an axis control command, be sure to use ZDO...ZEND even when a single statement is used.

If the workpiece coordinate on the first axis is greater than 30 and #RV[0] matches #RV[3], a movement on the A-axis starts.
//1 ZONCE [[#100101 GT 30.] AND [#RV[0] EQ #RV[3]]] ZDO ; **G91 G00 A20. ;** **ZEND ;**

Similarly, use ZDO...ZEND for a multi-statement including an axis control command.

If the workpiece coordinate on the second axis is equal to or less than 10, a movement on the V-axis starts and the Y1.0 signal is set to 1.
//1 ZONCE [#100102 LE 10.] ZDO ; **G91 G00 V10. ;** **#IOY[1,0] = 1 ;** **ZEND ;**

17.4.2 Condition Transition (ZEDGE Statement)

After ZEDGE, <conditional-expression> and <real-time-macro-statement> are coded.
// ZEDGE A B (If A becomes true, B is executed.)

- //(n) ZEDGE [<conditional-expression>] <real-time-macro-statement>

When <conditional-expression> makes a transition from false to true, an RTM statement specified after <conditional-expression> is executed. <conditional-expression> conforms to the custom macro specifications.
With the ZEDGE command, the result of the first evaluation of <conditional-expression> is always false.

Explanation

The difference between the ZONCE command and ZEDGE command is that if <conditional-expression> is false, the RTM statement itself is terminated in the case of ZONCE, while the evaluation of <conditional-expression> continues until <conditional-expression> becomes true in the case of ZEDGE. (With a one-shot real time command, the evaluation of <conditional-expression> continues until the execution of the NC statement that started operation simultaneously with the command ends.)

On the rising edge of the G address signal 4.3, the workpiece coordinate on the third axis is read.
// ZEDGE [#IOG[4,3] EQ 1] #RV[0]=#100103 ;

In the example above, even if the [#IOG[4,3] EQ 1] is true from the beginning, #RV[0]=#100103 of the RTM statement is not executed. #RV[0]=#100103 is executed when the result of evaluation of [#IOG[4,3] EQ 1] changes from false to true.

As in the case of ZONCE, multiple RTM statements can be coded in <real-time-macro-statement>.
If <conditional-expression-1> specifies an axis control command, be sure to use ZDO...ZEND even when a single statement is used.
Code the following by using ZDO...ZEND of multi-statement structure:
// ZEDGE [<conditional-expression>] ZDO ;
<real-time-macro-statement-1> ;

<real-time-macro-statement-2> ;
:
ZEND ;

On the falling edge of the X address signal, a movement on the B-axis is started and the Y1.0 signal is set to 1.
// ZEDGE [#IOX[1,3] EQ 0] ZDO ; **G91 G00 B10. ;** **#IOY[1,0] = 1 ;** **ZEND ;**

On the rising edge of the G address signal, a movement on the U-axis is started.
// ZEDGE [#IOG[4,3] EQ 1] ZDO ; **G91 G00 U25. ;** **ZEND ;**

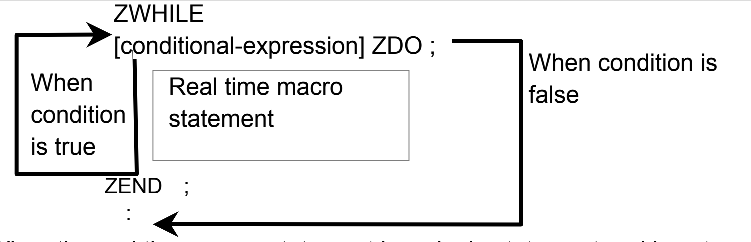
If the remaining travel distance on the second axis is equal to or less than 10, the rapid traverse override value is changed.
// ZEDGE [#100802 LE 10.] ZDO ; **#IOG[14,0]=0 ;** **#IOG[14,1]=1 ;** **ZEND ;**

17.4.3 Repetition (ZWHILE Statement)

After ZWHILE, a conditional statement is coded.
// ZWHILE A B (While A is true, B is executed repeatedly.)

- //(n) ZWHILE [<conditional-expression>] <real-time-macro-statement>

While <conditional-expression> is true, the RTM statement specified after <conditional-expression> is executed.
If <conditional-expression> is not satisfied, the ZWHILE statement is terminated, and the next block is processed.



* When the real time macro statement is a single statement and is not an axis control statement, ZDO and ZEND are not required.
* When “conditional-expression” is fulfilled and “Real time macro statement” is complete, interpolation cycle time has passed before the next evaluation of "ZWHILE conditional-expression".

Explanation

While <conditional-expression> is true, the command or commands between ZDO and ZEND after ZWHILE are executed.
If <conditional-expression> is not satisfied, the command after ZEND is processed.
The same conditional expression and operators as for the ZONCE statement are used.

While the F address 234.1 signal is 1, an incremental movement on the U-axis is repeatedly performed, and #RV[0] is incremented by 1 each time.
//1 ZWHILE [#IOF[234,1] EQ 1] ZDO ; G91 G00 U10. ;   #RV[0] = #RV[0]+1 ; ZEND ;

17.4.4 Multi-statement (ZDO...ZEND Statement)

- //(n) ZDO ;
<real-time-macro-statement-1> <real-time-macro-statement-2>
<real-time-macro-statement-3> …
ZEND ;

One or multiple statements enclosed in ZDO...ZEND are regarded as a one RTM statement (multiple-statement structure).

The maximum number of RTM statements in one multi-statement (between ZDO...ZEND) is subject to change (Because it depends on the contents of RTM statements).

ZDO...ZEND is used in the following cases:

- When multiple real time statements are handled as a series of commands

If the workpiece coordinate on the first axis exceeds 30, the rapid traverse override value is changed.
// ZEDGE [#100101 GT 30.] ZDO ; #IOG[14,0]=1 ; #IOG[14,1]=0 ; ZEND ;

- When an axis control command is specified

On the rising edge of the X signal 7.0, a movement on the V-axis is started.
// ZEDGE [#IOX[7,0] EQ 1] ZDO ; G91 G00 V35. ; ZEND ;

- Nesting

ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND cannot be nested and overlapped.
For details, see the following:

1. ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND may be used any number times.

```
// ZWHILE [···] ZDO ;
    Processing
ZEND ;
  :
// ZONCE[···] ZDO ;
    Processing
ZEND ;
  :
```

2. One ZDO...ZEND range must not overlap another ZDO...ZEND range.

```
// ZONCE [···] ZDO ;
    Processing
// ZWHILE [···] ZDO ;
  :
ZEND ;
    Processing
ZEND ;
```

3. ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND must not be nested.

```
// ZWHILE [···] ZDO ;
  :
  ZONCE [···] ZDO ;
    :
    ZEDGE [···] ZDO ;
      Processin
    ZEND ;
    :
  ZEND ;
  :
ZEND ;
```

- Endless loop

An endless loop is formed if the conditional expression enclosed in brackets after the ZWHILE statement is always satisfied.

Example)
In the example below, #RV[0] is counted up unconditionally.
// ZWHILE [1] #RV[1]=#RV[1]+1 ;

Sample program

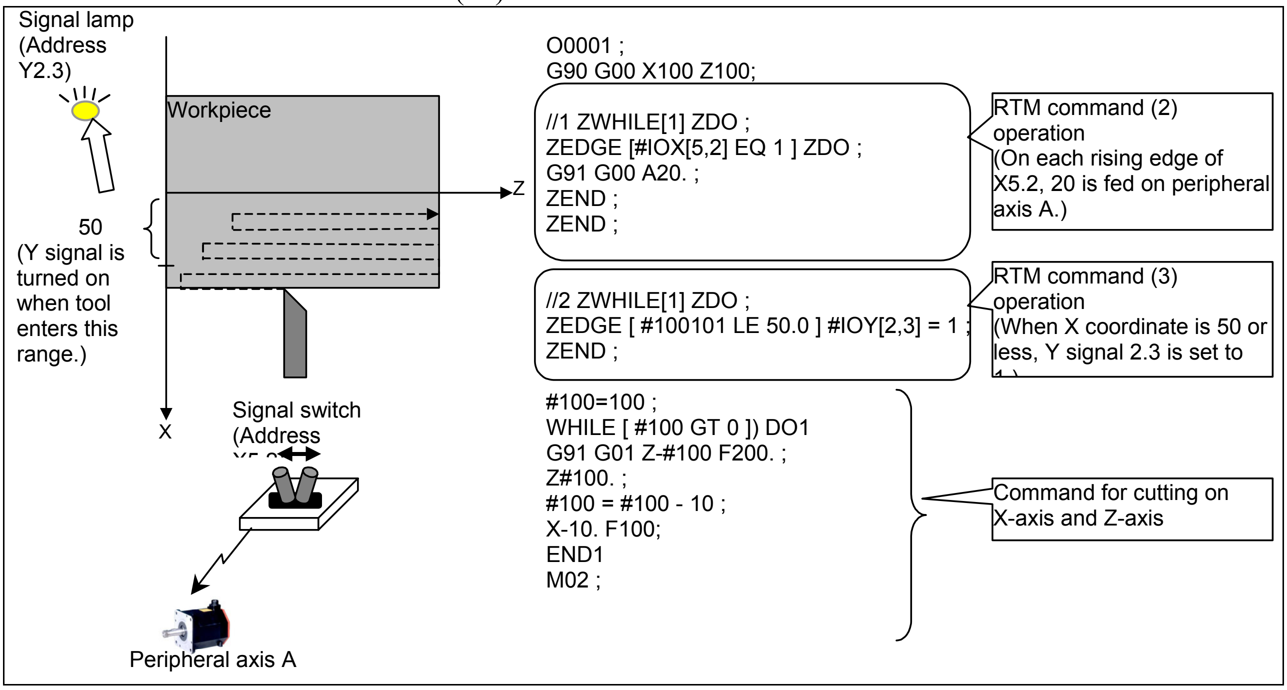
The sample program below exercises the following three control operations at the same time.

(1) A cutting operation is performed on the X-axis and Z-axis.

(2) On each rising edge of the X signal 5.2, 20 is fed on the peripheral axis A.

(3) When the workpiece coordinate on the X-axis (#100101) is equal to or less than 50, the Y signal 2.3 is set to 1.

The operation of (1) is coded in the main NC statement.
The operation of (2) is coded in the first modal RTM command (//1).
The operation of (3) is coded in the second modal RTM command (//2).

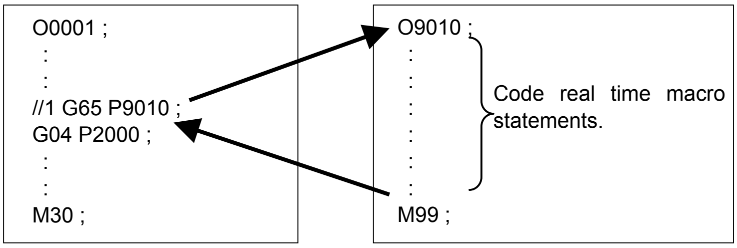


17.5 MACRO CALL

A series of RTM statements can be formed into a subprogram, which can be called from the main program.

When G65 is specified in an RTM command, the real time macro specified in address P is called.

G65 P p ;   P : Number of real time macro program to be called



The execution of real time macro //1 including O9010 starts simultaneously with the G04 block of main program O0001.

Explanation

- Calling

In address P after G65, specify the program number of a real time custom macro to be called.

NOTE

1. In argument P of G65, only a constant value can be used. No value can be specified through a variable.
   Example:
   // G65 P9010 ;   Correct
   // G65 P#RV[0] ;   Incorrect
2. The inclusion of another NC command (such as G01 X100.0 G65 Pp) is not allowed. If another NC command is included, PS0127 alarm is issued.
3. The G65 block for calling a real time macro does not make a single block stop.
4. On the other hand, a real time macro program called by real time macro calling makes a single block stop.

- Call destination real time program

In a called real time macro program, only an RTM statement can be coded.

In a called real time macro program, no additional RTM command may be executed. (The RTM command symbol ‘//’ may not be coded.) For example, do not execute the following program:

```
O0001 ;              O9010 ;
// G65 P9010 ;       // #RV[0]=1 ;
M02 ;                M99 ;
```

Code M99 in the last block only.
For example, do not execute the following program:

```
O0001 ;              O9010 ;
// G65 P9010 ;       ZEDGE [#RV[0] EQ 1] M99 ;
M02 ;                G91 G00 X50. ;
                     M99 ;
```

When executing an RTM statement in a subprogram, code an NC statement before the M99 block for returning to the calling program.

```
O0001 ;              O9010 ;
G65 P9010 ;          // #RV[0] = 20 ; Real time macro statement
M02 ;                G04 ;            NC statement
                     M99 ;
```

- Format

Specify G65 at the start of a block.

- Call nesting level

Macro call nesting is not allowed.

- Differences from macro call using a custom macro

Macro call using a real time custom macro and macro call using a custom macro differs as described below.

- In macro call using a custom macro, an argument (data to be passed to a macro) and the number of repetitions can be specified. In macro call using a real time custom macro, such information cannot be specified.
- With a real time custom macro, other types of macro call (macro call using G66, G66.1, G, and M code) and subprogram call are not allowed.

17.6 OTHERS

If an axis control command is followed by a macro command in an RTM command, the execution of the macro command starts when the axis control command is completed or deceleration starts.
If deceleration on the X-axis starts upon completion of distribution according to the axis control command of (1), for example, the macro command of (2) is executed. If acceleration/deceleration is not applied on the X-axis, (2) is executed upon completion of distribution on the X-axis.

```
// ZDO ;
G91 G00 X30 ;     (1) Axis control command of the RTM statement
#RV[0] = 1 ;      (2) Macro command of the RTM statement
ZEND ;
```

17.7 AXIS CONTROL COMMAND

In an RTM statement, an M code and G code for specifying a movement can be specified. For axis control, the PMC axis control interface is used. The specifications differ from the specifications for the G and M codes used with an NC statement.

Format

// ZDO ;
G___ IP ___F___ ;
:
ZEND ;

For an axis control command, ZDO...ZEND of multi-statement structure is used for coding even if a single statement (single RTM statement) is used.

> ⚠ **CAUTION**
> Do not specify an NC statement for an axis to be controlled with an RTM statement. Moreover, do not exercise axis control with an RTM statement on an axis to be controlled with an NC statement.

> **NOTE**
> In one block, only one axis may be specified. Do not specify two or more axes in the same block.

Explanation

- Setting

For an axis control command in an RTM statement, the interface of a PMC axis control group set by a parameter beforehand is used.
For an axis to be controlled with an RTM statement, set a group to be used, with parameter No. 8010. Then, set bit 0 (XRT) of parameter No. 8011 for the axis to 1.

> **NOTE**
> An axis for which bit 0 (XRT) of parameter No. 8011 is set to 1 is dedicated to real time custom macros, so that such an axis cannot be used with PMC axis control.

- Relationship with PMC axis control

Axis control based on an RTM statement uses the PMC axis control interface. So, the specifications for a move command in each block within an RTM statement are generally equivalent to the specifications for the PMC axis control command.
For the detailed specifications and restrictions related to axis control, refer to the specifications of PMC axis control as well.

> **NOTE**
> 1 An axis used with an RTM statement must not be specified from PMC axis control.
> 2 A PMC axis control group used with an RTM statement must not be specified from PMC axis control.

- Operation command code

The table below indicates the G codes that can be specified in an RTM statement.
The ◤ mark in the table indicates modal G codes when an RTM command starts.

Code	Group	Meaning
◤G00	01	Positioning
G01		Feed at specified feedrate
G28	00	Reference position return
G53		Machine coordinate system selection
G90	03	Absolute command NOTE 1
◤G91		Incremental command NOTE 2
◤G94	05	Feed per minute
G95		Feed per revolution

> **NOTE**
> 1 The absolute command (G90) is valid for machine coordinate system selection (G53) only.
> 2 The incremental command (G91) is valid for positioning (G00), linear interpolation (G01), and reference position return (G28) only.
> 3 In T series G code system A as well, not G98/G99 but G90/G91 are valid with an RTM command.

- Modal information

Basically, modal information such as F and modal G codes in an RTM statement is independent in an NC statement and each RTM statement. Modal information in an RTM statement is initialized when the execution of the RTM statement is started (when a block specified with // is executed).
The initial value of modal information in an RTM statement is as follows:

State of the G codes marked with ◤ in the operation command code list
F code : F0

> ⚠ **CAUTION**
> With the G codes (inch input/metric input) of group 06, the same information as the modal information of an NC statement is used in an RTM statement. Do not change the modal information of group 06 with an NC statement in a block after the first RTM statement coded in a program.

> **NOTE**
> With an RTM command, bit 0 (G01), bit 3 (G91), and bit 4 (FPM) of parameter No. 3402 are invalid when the power is turned on or in the cleared state.

Example 1)

Modal information can be controlled independently in an NC statement and each RTM statement.

```
O0001 ;
G90 G01 X100. Y100. F500. ;       (1)
//1 ZDO ;
Z50. ;                            (2)
G01 Z100. F100. ;                 (3)
ZEND ;
//2 ZDO ;
G01 A30. F200. ;                  (4)
A50. ;                            (5)
ZEND ;
X200. ;                           (6)
:
```

The modal information of command (2) is G00, G91, and G94, regardless of the modal information (G90 G01 set in (1)) of the NC statement.

The modal information of command (5) is G01, G91, G94, and F200.. This modal information is not affected by the modal information (command (1)) of the NC statement and the modal information (commands (2) and (3)) in the RTM statement with modal ID=1.

The modal information of command (6) is not affected by the modal information (commands (2) to (5)) in the RTM statement, so that the modal information is G01, G90, and F500..

Example 2)

Modal information is initialized when the execution of each RTM command is started. Even if the same program includes RTM commands with the same ID, the modal information of the RTM command executed first is not inherited by the RTM command executed next.

```
O0001 ;
G90 G01 X100. Y100. F500. ;
//1 ZDO ;
G01 Z100. F3000. ;                (1)
```

```
ZEND ;
X200. ;
//1 ZDO ;
Z200. ;                                   (2)
ZEND ;
X300. ;
:
```

The modal information of command (2) is G91 and G00, regardless of command (1).

- Single block stop

If an NC statement is placed in the single block stop state, for example, by the single block stop signal SBK, the axis control command of an RTM statement also comes to a single block stop.
However, the automatic operation start signal STL is turned off when all conditions on the NC statement side are satisfied, even if the axis control command of an RTM statement is not terminated.
If an RTM command consists of multiple statements and an axis control command is coded in multiple blocks, only the block of the RTM statement that is currently executing an axis command can be brought to a single block stop by setting the group-by-group single block stop signal (ESBKg) for PMC axis control corresponding to the axis to 1.

- Feed hold

Even when an NC statement enters the automatic operation halt state, the axis control command of an RTM statement does not stop immediately but stops at the time of termination of the block currently being executed. To stop also the axis control command of an RTM statement immediately when the NC statement enters the automatic operation halt state, control the temporary stop signal (ESTPg) for PMC axis control of the related group by monitoring the automatic operation halt in-progress signal (SPL).

- Reset

Even when the CNC is reset by an MDI reset, the external reset signal (ERS), or the reset and rewind signal (RRW), the axis control command of an RTM statement does not stop immediately but stops at the time of termination of the block currently being executed. When the emergency stop state is set, however, the RTM statement also stops immediately.
During execution of the axis control command of an RTM statement, the real time macro being executed can be stopped independently of the NC statement by setting the group-by-group reset signal (ECLRg) for PMC axis control corresponding to the axis to 1. To stop also the axis control command of an RTM statement when the CNC is reset, control the reset signal (ECLRg) for PMC axis control of the related group by monitoring the reset signal (RST).

- Alarm stop

Even when a P/S alarm is issued with an NC statement, the axis control command of the RTM statement being executed does not stop

immediately but stops at the time of termination of the block currently being executed. Moreover, even when an overtravel alarm is issued for an axis other than the axis controlled by the RTM statement being executed, the RTM statement being executed does not stop immediately but stops at the time of termination of the block currently being executed.

- Alarm reset

When the group-by-group alarm signal (EIALg) for PMC axis control corresponding to the axis specified by an RTM statement is set to 1, correct the cause of the alarm, then reset the CNC, and reset PMC axis control with the reset signal (ECLRg) for PMC axis control of the related group.
(Tip)

In the following cases, the alarm signal (EIALg) is set to 1:

(1) When a servo alarm is issued
(2) When an overtravel alarm is issued
(3) When P/S0130 alarm is issued
(4) When P/S0139 alarm is issued

- Interlock

In an RTM statement, the interlock signal (*IT) for an NC statement, interlock signal for each axis (*ITn), interlock signal for each axis direction (+ITn/-ITn) are invalid. Instead, the axis control temporary stop signal (ESTPg) for PMC axis control is valid in an RTM statement. When temporarily stopping the axis controlled by an RTM statement, control the corresponding axis control temporary stop signal (ESTPg) for PMC axis control.

The cutting block start interlock signal *CSL (G8.1) and the block start interlock signal *BSL (G8.3) are invalid.

- Machine lock

The same machine lock signals (all axes/each axis) as used with an NC statement are used. However, by disabling machine lock for PMC axis control with the following parameters, machine lock can be disabled for the axis being controlled by an RTM statement:

Bit 0 (MLE) of parameter No. 8001
Bit 1 (MLS) of parameter No. 8006

- Dry run

With bit 2 (OVE) of parameter No. 8001, whether to use the dry run signal (DRN) for an NC statement or the dry run signal (EDRN) for a PMC axis can be chosen. When enabling a dry run feedrate for rapid traverse in an RTM statement, set bit 3 (RDE) of parameter No. 8001 to 1. If a feed command specifying a feedrate is specified when the manual rapid traverse selection signal (RT or ERT) is set to 1, the cutting feedrate is used as the feedrate during dry run. ("Dry run feedrate × maximum manual feedrate override value" is not applicable.)

- In-position check

In the in-position state, the in-position signal (EINPg) is set to 1.
When bit 6 (NCI) of parameter No. 8004 is set to 1, no in-position check is made during axis control based on an RTM statement. The setting of bit 5 (NCI) of parameter No. 1601 for an NC statement is invalid. (However, the in-position signal (INPx) for an NC statement is affected by the value of NCI.)

- Manual absolute

The manual absolute signal (*ABSM) is invalid.

- External deceleration

By setting bit 0 (EDC) of parameter No. 8005 to 1, the external deceleration function can be used also for the axis being controlled by an RTM statement. Note, however, that the external deceleration function is unusable for feed with a specified feedrate (feed per revolution). For an external deceleration rate, the external deceleration rate setting for rapid traverse (parameter No. 1427, No. 1441, and No. 1444) is valid, regardless of the type of feed.
The external deceleration function for the axis being controlled by an RTM statement is also valid for feed with a specified feedrate (feed per minute), regardless of the settings of bit 4 (EDP) of parameter No. 1005 and bit 5 (EDM) of parameter No. 1005. (The enabling of rapid traverse only is impossible.)

- Mirror image

When enabling mirror image for the axis being controlled by an RTM statement, set bit 0 (EMR) of parameter No. 8008 to 1, and set either bit 0 (MIRx) of parameter No.0012 or mirror image signal MIx to '1'.
Programmable mirror image is not available.

NOTE

1. The same group as the PMC axis control group used for axis control cannot be executed with an RTM statement.
2. No NC statement can specify an axis being controlled by an RTM statement.

NOTE

3 An alarm is issued if, during execution of an RTM statement, an attempt is made to execute another RTM statement with the same ID. In the program below, for example, the RTM statement of (1) operates using the NC statement of (2) as a trigger, and the RTM statement of (3) operates using the NC statement of (4) as a trigger. An alarm is issued if (1) is still operating when (4) is started after completion of the interpolation of (2).

```
//1 ZDO ;
G91 G00 A100. ;   (1)  RTM statement
ZEND ;
G01 X30. Y20. ;   (2)  NC statement
//1 ZDO ;
G91 G00 B100. ;   (3)  RTM statement
ZEND ;
X40. Y50. ;       (4)  NC statement
```

4 When T series G code system A is used, address words (such as U, W, H, and V) for incremental commands must not be coded. Otherwise, P/S alarm 413 is issued.

Command details

- Rapid traverse

A movement is made at a rapid traverse rate on an axis from the current position to the point separated by a specified value.

Format

// ZDO ;
G91 G00 IP ___ ;
ZEND ;

G91 : G code for incremental command
IP __: Travel distance

NOTE

1 Only one axis can be specified in one block.
2 The absolute command (G90) cannot be specified.
3 The rapid traverse overlap function cannot be used.
4 When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.

- Rapid traverse override
  With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal (ROV) for an NC statement or the rapid traverse override signal (EROV) dedicated to PMC axis control can be chosen.

NOTE

Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

- Feed with a specified feedrate (feed per minute)

A movement is made at a feedrate specified in F on an axis from the current position to the point separated by a specified value.

Format

// ZDO ;
G94 G91 G01 IP ___ F___ ;
ZEND ;

G94 : G code for feed per minute
G91 : G code for incremental command
IP __: Travel distance
F__ : Feedrate command (mm/min or inch/min)

NOTE

1 Only one axis can be specified in one block.
2 The absolute command (G90) cannot be specified.
3 The block overlap function cannot be used.
4 When IS-A is used, a feedrate below 10 mm/min is discarded.
5 The feedrate cannot be clamped to a parameter-set maximum cutting feedrate.
6 No time is added to the cumulative cutting time.
7 Even if this command is executed, the cutting in-progress signal is not output.
8 The operation between blocks for feed with a specified feedrate is performed in the cutting mode.
9 For this command, the feedrate arrival signal SAR (G029.4) cannot be used.

- Feedrate specification range
  The table below indicates the feedrate specification range.

	Linear axis		Rotation axis (deg/min)
	Metric input (mm/min)	Inch input (inch/min)	
IS-A	10. to 240000.	0.1 to 6553.5	10. to 240000.
IS-B	1. to 65535.	0.01 to 655.35	1. to 65535.
IS-C	0.1 to 6553.5	0.001 to 65.535	0.1 to 6553.5
IS-D	0.01 to 655.35	0.0001 to 6.5535	0.01 to 655.35
IS-E	0.001 to 65.535	0.00001 to 0.65535	0.001 to 65.535

> **NOTE**
> 1 Be sure to set the following parameters to 0:
> F10 (bit 3 of parameter No. 8002)
> EFD (bit 4 of parameter No. 8006)
> PF1 (bit 4 of parameter No. 8002)
> PF2 (bit 5 of parameter No. 8002)
> When a value other than 0 is set, the feedrate specification range changes, depending on the set parameter.
> 2 A value less than the lowest feedrates indicated above is discarded.

- Feedrate override
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override signal (*FV) for an NC statement or the feedrate override signal (*EFV) dedicated to PMC axis control can be chosen.

> **NOTE**
> 1 The second feedrate override function cannot be used.
> 2 The feedrate override function cannot be disabled using #3004.

- Override cancel
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override cancel signal (OVC) for an NC statement or the feedrate override cancel signal (EOVC) dedicated to PMC axis control can be chosen.

- Operation between blocks
  If feed with a specified feedrate is performed in succession in RTM statements, no stop occurs between blocks, but the next block is executed. With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate.

> **NOTE**
> 1 The error detect signal (SMZ) cannot be used.
> 2 With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate for feed on the same axis only.

- Acceleration/deceleration time constant
  For an acceleration/deceleration time constant to be used for feed with a specified feedrate in an RTM statement when exponential acceleration/deceleration is used, whether to use the time

constant for an NC statement or the time constant dedicated to PMC axis control can be chosen using parameter No. 8030.

> **NOTE**
> Look-ahead acceleration/deceleration before interpolation cannot be used.

- Feed with a specified feedrate (feed per revolution)

A movement is made at a feedrate specified in F on an axis from the current position to the point separated by a specified value.

Format

// ZDO ;
G95 G91 G01 IP ___ F___ ;
ZEND ;

G95 : G code for feed per revolution
G91 : G code for incremental command
IP __: Travel distance
F__ : Feedrate command (mm/rev or inch/rev)

> **NOTE**
> 1 Only one axis can be specified in one block.
> 2 The absolute command (G90) cannot be specified.
> 3 The block overlap function cannot be used.
> 4 Be sure to set the parameters below to 0. If a value other than 0 is set, the feedrate specification range changes, depending on the set parameter.
> FR1 (bit 6 of parameter No. 8002)
> FR2 (bit 7 of parameter No. 8006)
> 5 The feedrate is clamped to the maximum cutting feedrate set in parameter No. 8022. (Parameter No. 1430 for an NC statement is not valid for this command.)
> 6 To use feed per revolution, a position coder is required at all times. (Bit 0 (NPC) of parameter No. 1402 is not applicable to this command.)
> 7 No time is added to the cumulative cutting time.
> 8 Even if this command is executed, the cutting in-progress signal is not output.
> 9 The operation between blocks for feed with a specified feedrate is performed in the cutting mode at all times.
> 10 For this command, the feedrate arrival signal SAR (G029.4) cannot be used.

- Feedrate specification range
  The table below indicates the feedrate specification range.

	Linear axis		Rotation axis (deg/rev)
	Metric input (mm/rev)	**Inch input (inch/rev)**	
T series	0.001 to 65.535	0.000001 to 0.65535	0.001 to 65.535
M series	0.01 to 500.00	0.0001 to 6.5535	0.01 to 500.00

- Feedrate override
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override signal (*FV) for an NC statement or the feedrate override signal (*EFV) dedicated to PMC axis control can be chosen.

NOTE
1. The second feedrate override function cannot be used.
2. The feedrate override function cannot be disabled using #3004.

- Override cancel
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override cancel signal (OVC) for an NC statement or the feedrate override cancel signal (EOVC) dedicated to PMC axis control can be chosen.

- Operation between blocks
  If feed with a specified feedrate is performed in succession in RTM statements, no stop occurs between blocks, but the next block is executed. With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate.

NOTE
1. The error detect signal (SMZ) cannot be used.
2. With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate for feed on the same axis only.

- Acceleration/deceleration time constant
  For an acceleration/deceleration time constant to be used for feed with a specified feedrate in an RTM statement when exponential acceleration/deceleration is used, whether to use the time constant for an NC statement or the time constant dedicated to PMC axis control can be chosen using parameter No. 8030.

NOTE
Look-ahead acceleration/deceleration before interpolation cannot be used.

- Reference position return

A movement is made at the rapid traverse rate to the first reference position on a specified axis. Upon completion of reference position return, the return completion lamp is turned on.

Format

// ZDO ;
G91 G28 IP 0 ;
ZEND ;

G91 : G code for incremental command
IP 0 : Specify an axis on which reference position return is to be performed.
Be sure to specify 0 as the travel distance.

NOTE

1. Only one axis can be specified in one block.
2. No intermediate point may be specified. (Be sure to specify 0 as the travel distance. Otherwise, P/S alarm 410 is issued.) If a movement needs to be once made to an intermediate point, specify positioning with rapid traverse (G00)/machine coordinate system selection (G53) then specify this command.
3. The absolute command (G90) cannot be specified.
4. When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.
5. Before this command can be specified, a reference position must be established. (With this command, do not specify an axis on which a reference position is not established.)
6. Before specifying this command with the T series, cancel tool offset and tool-nose radius compensation. Before specifying this command with the M series, cancel cutter compensation, tool length compensation, and tool offset.

- Rapid traverse override

With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal for an NC statement or the rapid traverse override signal dedicated to PMC axis control can be chosen.

NOTE

Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

- Machine coordinate system selection

When a position in the machine coordinate system is specified, a movement is made to the position on the axis by rapid traverse. The G53 code for machine coordinate system selection is a one-shot G code, so that a command in the machine coordinate system is valid in the block that specifies G53. When moving the tool to a machine-specific position such as a tool change position, perform programming in the machine coordinate system based on G53.

Format

// ZDO ;
G90 G53 IP ___ ;
ZEND ;

G90 : G code for absolute command
IP __: Position in machine coordinate system

NOTE

1 Only one axis can be specified in one block.
2 The incremental command (G91) cannot be specified.
3 When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.
4 Before this command can be specified, a reference position must be established. (With this command, do not specify an axis on which a reference position is not established.)
5 Before specifying this command with the T series, cancel tool offset and tool-nose radius compensation. Before specifying this command with the M series, cancel cutter compensation, tool length compensation, and tool offset.

- Rapid traverse override

With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal for an NC statement or the rapid traverse override signal dedicated to PMC axis control can be chosen.

NOTE

Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

17.8 NOTES

- Address without the decimal point

In general, an NC address without the decimal point is subject to calculator-type decimal point input when bit 0 (DPI) of parameter No. 3401 or bit 0 (AXDx) of parameter No. 3455 is set to 1. In other cases, an NC address without the decimal point is regarded as a least input increment.

- Indirect axis address specification

When indirect axis address specification is used, use a direct reference argument. No indirect reference may be used.

Example)

AX[1]	Correct
AX[#RV[0]]	Incorrect

- Diameter specification/radius specification

For an axis to be controlled with an RTM statement, select radius specification. (Set bit 3 (DIA) of parameter No. 1006 to 0.)

- P/S alarm 224

If bit 0 (ZRN) of parameter No. 1005 is set to 0, and an axis control command is specified in an NC statement before a reference position return operation is not performed even once after power-up, P/S alarm 224 (ZERO RETURN NOT FINISHED.) is issued. In axis control using an RTM statement, the alarm is not issued even when bit 0 (ZRN) of parameter No. 1005 is set to 0, unless bit 6 (EZR) of parameter No. 8006 is set to 1.

- Axis removal

Even when axes are removed, interlock is not applied to the axis being controlled by an RTM statement.

- Stroke limit check before movement

No stroke limit check before movement is made with a block in an RTM statement.

- Others

For the axis being controlled by an RTM statement, the functions disabled for PMC axis control cannot be used.

- Other disabled functions

For the axis being controlled by an RTM statement, the functions listed below as examples are disabled.

Do not specify these functions for the axis being controlled by an RTM statement.

- Feed-forward
- Advanced preview feed-forward
- Fine acceleration/deceleration
- Automatic corner deceleration

- One-digit F code feed
- Scaling
- Coordinate system rotation
- Polar coordinate interpolation
- Balance cutting
- Feed stop
- Constant surface speed control
- Positioning function based on optimal acceleration, etc.

⚠ **CAUTION**

In an RTM statement, do not specify an axis used with any of the following functions:

- Feed axis synchronization control
  Be sure to set the simple synchronous axis selection signal (SYNCn) and simple synchronous manual feed axis selection signal (SYNCJn) of the axis to 0.
- Axis recomposition
- Index table indexing
- Three-dimensional coordinate system conversion
- Rigid tapping
- Compound machining
- Polygon turning
- Cs contour control
- AI contour control I/II

17.9 LIMITATION

Major general notes on RTM commands are provided below.

- Background drawing

The RTM command has no effect in background drawing. Do not specify an RTM command during background drawing.

- Interrupt-type custom macro

In an interrupt-type custom macro, no RTM command can be coded.

- Macro executor

In a macro executor, no RTM command can be coded.
Moreover, no macro executor can be coded from an RTM command.
In a series of programs, do not code a macro executor (execution macro) and RTM command at the same time.
In addition, observe the following:

- Do not code an RTM command in a program that uses an execution macro.
- Do not call an execution macro from a program with an RTM command coded.
- Do not code an RTM command in an execution macro.
- Do not use an execution macro as an RTM command trigger.

- Sequence number

No sequence number can be coded in an RTM command. No RTM command can be searched for a sequence number.

- Optional block skip

Optional block skip is unusable.
A slash (/) that appears in the middle of <expression> (enclosed in [ ] on the right-hand side of an arithmetic/logical expression) is regarded as a division operator; it is not regarded as an optional block skip code.

- Single block

When an RTM command is used, a single block stop generally occurs according to an NC statement, regardless of the values of bit 5 (SBM) of parameter No. 6000 and bit 7 (SBV) of parameter No. 6000. However, no single block stop occurs according to G65 for real time macro call.

- Processing

If there are many RTM statements between NC statements, the command flow may break.

- DNC operation

DNC operation is disabled.

- Operation in each event

If an event such as an emergency stop or alarm occurs during execution of an RTM command, the NC command and RTM command generally operate as indicated below.

Event	NC command	RTM command consisting of a macro command	RTM command including an axis control command
P/S alarm issued in NC command execution	Operation stops when the block involving the P/S alarm cause starts.	The RTM command stops when the NC command stops.	If the RTM statement being executed when the NC command stops is an axis control command, operation on the axis stops when the block stops.
Alarm issued in RTM command execution	Operation stops with the alarm.	The RTM statement stops at that point. Other RTM commands stop when the NC command stops.	The RTM statement stops at that point. Other RTM commands stop operation on the axis when axis control by the RTM statement being executed when the NC statement stops ends.
Single block (SBK signal)	Operation stops when the command being executed ends.	The RTM command is suspended when the NC command stops. When the NC command restarts, the RTM command restarts.	If the RTM statement being executed when the NC command stops is an axis control statement, operation on the axis stops when the block ends.
Feed hold (*SP signal)	A gradual stop occurs.	The RTM command stops when the NC command stops. When the NC command restarts, the RTM command restarts.	If the RTM statement being executed when the NC command stops is an axis control statement, operation on the axis stops when the block ends.
NC command stopped by a reset (ERS signal)	A gradual stop occurs, and the command also ends.	The RTM command stops when the NC command stops.	If the RTM statement being executed when the NC command stops is an axis control command, operation on the axis stops when the block ends.
Emergency stop	An immediate stop occurs, and the command also ends.	The command immediately ends.	Operation on the axis immediately stops, and the command also immediately ends.
End of the NC command	-	If the RTM command being executed is a one-shot RTM command, the command ends. If the RTM command being executed is a modal RTM command, the command continues execution.	If the one-shot RTM command being executed when the NC command ends is an axis control command, operation on the axis stops when the block ends.
Machine lock (MLK signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.
Dry run (DRN signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.
Interlock (*IT signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.

18 PROGRAMMABLE PARAMETER INPUT (G10)

Overview

The values of parameters and pitch error compensation data can be entered in a program. This function is used for setting pitch error compensation data when attachments are changed or the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.

Format

- Prameter entry mode

G10L52 ; Parameter entry mode setting
N_R_ ; For parameters other than the axis type
N_P_R_ ; For axis type parameters
:
G11 ; Parameter entry mode cancel

N_ : Parameter number
R_ : Parameter setting value (Leading zeros can be omitted.)
P_ : Axis number 1 to maximum controlled axis number (to be specified when an axis type parameter or spindle type parameter is specified)

NOTE
G10L52 cannot be used to enter pitch error compensation data.

- Pitch error compensation data entry mode

G10L50 ; Pitch error compensation data entry mode setting
N_R_ ; Pitch error compensation data entry
N_P_R_ ; For axis type parameters
:
G11 ; Pitch error compensation data entry mode cancel

N_ : Compensation position number for pitch errors compensation +10,000
R_ : Pitch error compensation data

NOTE
G10L50 cannot be used to enter parameter.

Explanation

- Setting value (R_)

Do not use a decimal point in the setting (R_) of a parameter or pitch error compensation data. As the value of R, a custom macro variable can be used.
When a parameter of real type is used, set an integer value in (R_) according to the increment system of the parameter.

- Axis number (P_)

As the axis number (P_), specify the order of a controlled axis to be displayed on the CNC display screen, by using an axis type parameter. For example, specify P2 for the control axis which is displayed second.

⚠ WARNING
1 Do not fail to perform reference position return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
2 The canned cycle mode must be cancelled before entering of parameters. When not cancelled, the drilling motion may be activated.

⚠ CAUTION
Compatibility with the Series 16*i*/18*i*/21*i*:
This model has parameters that are not compatible with the Series 16*i*/18*i*/21*i*. So, before using this function, make a check according to the parameter manual (B-63950EN) of this model.

NOTE
Other NC statements cannot be specified while in parameter input mode.

Example

1. Set bit 2 (SBP) of bit type parameter No. 3404

G10L52 ;	Parameter entry mode
N3404 R 00000100 ;	SBP setting
G11 ;	Cancel parameter entry mode

2. Change the values for the Z-axis (3rd axis) and A-axis (4th axis) in axis type parameter No. 1322 (the coordinates of stored stroke limit 2 in the positive direction for each axis).
   (When the increment systems for the 3rd and 4th axes are IS-B and millimeter machine, respectively)

G10L52 ;	Parameter entry mode
N1322P3R4500 ;	Change the value for the Z-axis to 4.500
N1322PP4R12000 ;	Change the value for the A-axis to 12.000
G11 ;	Cancel parameter entry mode

3. Change compensation point numbers 10 and 20 of pitch error compensation.

G10L50 ;	Pitch error compensation data entry mode
N10010R1 ;	Change the compensation point number from 10 to 1
N10020R5 ;	Change the compensation point number from 20 to 5
G11 ;	Pitch error compensation data entry mode

19 HIGH-SPEED CUTTING FUNCTIONS

19.1 AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II (G05.1)

Overview

The AI contour control I and AI contour control II functions are provided for high-speed, high-precision machining. This function enables suppression of acceleration/deceleration delays and servo delays that become larger with increases in the feedrate and reduction of machining profile errors.
There are two types of AI contour control; these two types are referred to temporarily as AI contour control I and AI contour control II. AI contour control I is designed mainly for part machining, and AI contour control II is for machining of successive minute straight lines such as mold machining and for machining by curve commands in NURBS and so on.
When a high-speed processing option is used with AI contour control II, high-speed operation processing is enabled, which reduces the interpolation cycle, therefore, faster and more precise machining is enabled. In addition, an option to expand read-ahead blocks to up to 1000 blocks is available.
In the descriptions below that are common to AI contour control I and AI contour control II, the term "AI contour control" is used.

Format

G05.1 Q_ ;
Q1 : AI contour control mode on
Q0 : AI contour control mode off

NOTE
1 Always specify G05.1 in an independent block.
2 The AI contour control mode is also canceled by a reset.
3 The AI contour control mode can be turned on at the start of automatic operation by setting bit 0 (SHP) of parameter No. 1604.

The AI contour control mode can be controlled also with the formats that have been used for the conventional advanced preview control, high-precision contour control, and AI high-precision contour control functions.

G08 P_ ;
P1 : AI contour control mode on
P0 : AI contour control mode off
G05 P_ ;
P10000 : AI contour control mode on
P0 : AI contour control mode off

NOTE
1 Always specify G08 and G05 in an independent block.
2 G05 can be specified only for AI contour control II.
3 The AI contour control mode is also canceled by a reset.
4 Valid functions are limited depending on the command format. For details, see the description of "Valid functions".

Valid functions

The functions listed below are valid in the AI contour control mode.

Valid functions are limited depending on the command format and whether AI contour control I or II is used.

Function	AI contour control I	AI contour control II	AI contour control II with high-speed processing
Number of blocks read ahead	30 (When G8 is specified: 1)	200 (When G8 is specified: 1)	600 (*1) (When G8 is specified: 1)
Look-ahead acceleration/deceleration before interpolation	Linear or bell-shaped acceleration/deceleration	Linear or bell-shaped acceleration/deceleration	Linear or bell-shaped acceleration/deceleration
Speed control with feedrate difference on each axis	Enabled	Enabled	Enabled
Speed control with acceleration in circular interpolation	Enabled	Enabled	Enabled
Speed control with acceleration on each axis	Enabled (When G8 is specified: Not enabled)	Enabled (When G8 is specified: Not enabled)	Enabled (When G8 is specified: Not enabled)
Smooth speed control	Not enabled	Enabled (When G8 is specified: Not enabled)	Enabled (When G8 is specified: Not enabled)
Speed control with cutting load	Not enabled	Enabled	Enabled
Disregard of feedrate command	Not enabled	Enabled	Enabled

*1 The number of blocks can optionally be expanded to 1000 blocks.

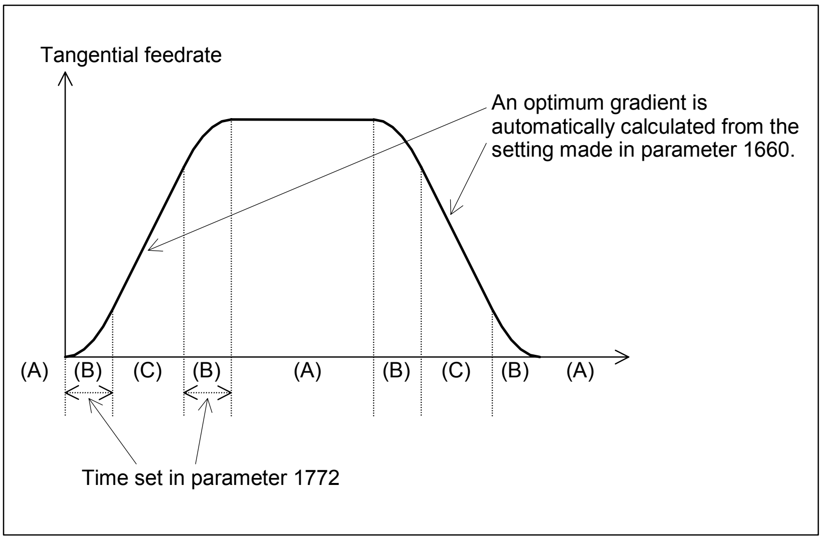
Explanation

- Look-ahead acceleration/deceleration before interpolation

There are two types of look-ahead acceleration/deceleration before interpolation, the linear acceleration/deceleration type and the bell-shaped acceleration/deceleration type.
Look-ahead bell-shaped acceleration/deceleration before interpolation produces smoother acceleration/deceleration.

- Setting an acceleration

A permissible acceleration for the linear acceleration/deceleration of each axis is set in parameter 1660. For bell-shaped acceleration/deceleration, acceleration change time (B) (period of transition from constant speed state (A) to constant acceleration/ deceleration state (C)) is set in parameter 1772. In the constant acceleration/deceleration state (C), acceleration/deceleration is performed with the maximum tangential acceleration not exceeding the permissible acceleration of each axis specified in parameter 1660. The acceleration change time specified in parameter 1772 is held constant, regardless of the tangential acceleration.



- Method of determining the tangent acceleration

Acceleration/deceleration is performed with the largest tangent acceleration/deceleration that does not exceed the acceleration set for each axis.

(Example)

X-axis permissible acceleration: 1000 mm/sec^2

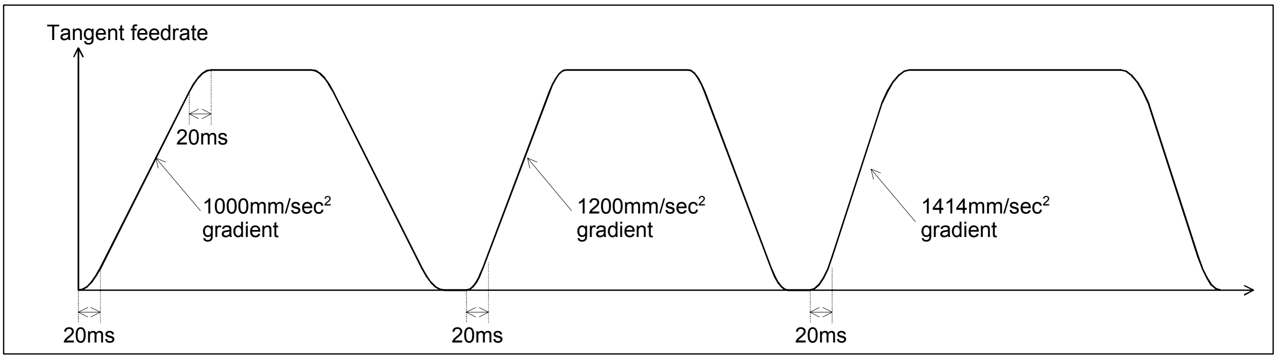
Y-axis permissible acceleration: 1200 mm/sec^2

Acceleration change time: 20 msec

Program:

N1 G01 G91 X20. F6000	Move on the X-axis.
G04 X0.01	
N2 Y20.	Move on the Y-axis.
G04 X0.01	
N3 X20. Y20.	Move in the XY direction (at 45 degrees).

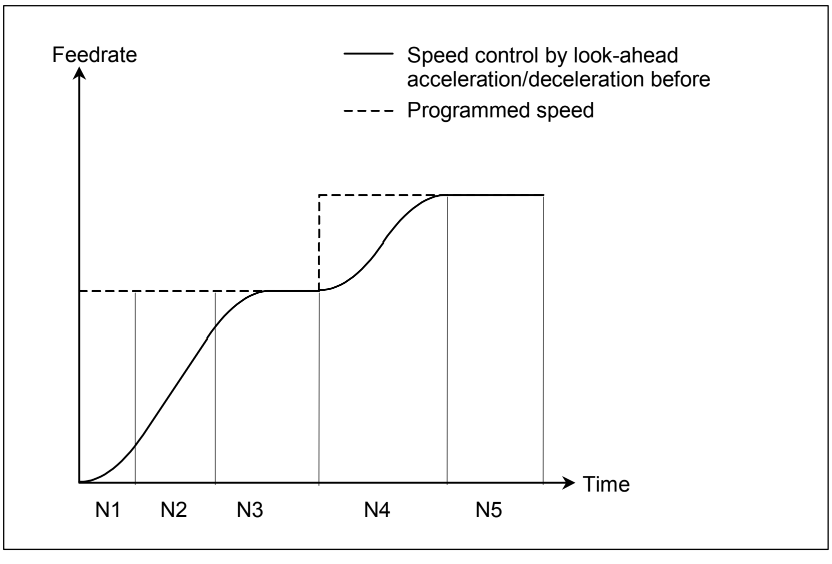
The acceleration in N3 is 1414 mm/sec^2. At this point, the acceleration on the X-axis is equal to the set value (1000 mm/sec^2).



- Acceleration

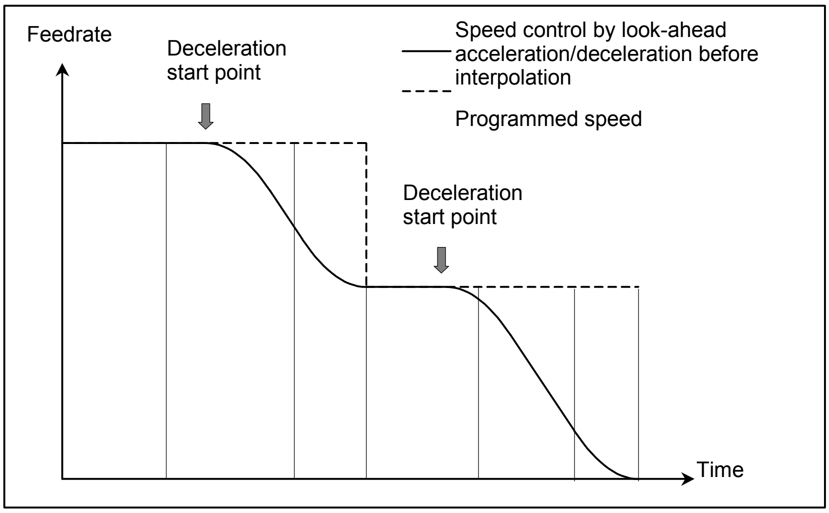
Acceleration is performed so that the feedrate programmed for a block is attained at the beginning of the block.

Acceleration can be performed over several blocks.



- Deceleration

Deceleration starts in advance so that the feedrate programmed for a block is attained at the beginning of the block.
Deceleration can be performed over several blocks.

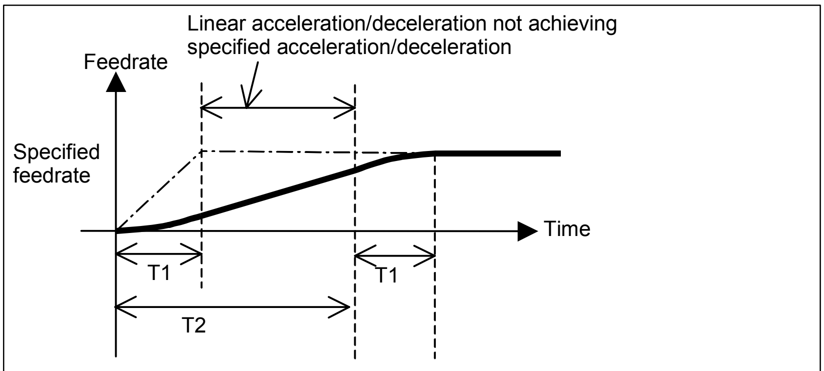


- Deceleration based on a distance

If the total distance of the blocks read ahead becomes shorter than or equal to the deceleration distance obtained from the current feedrate, deceleration starts.
If the total distance of the blocks read ahead during deceleration increases, acceleration is performed.
If the blocks of a small amount of travel are successively specified, deceleration and acceleration may be performed alternately, making the feedrate inconsistent.
To avoid this, decrease the programmed feedrate.

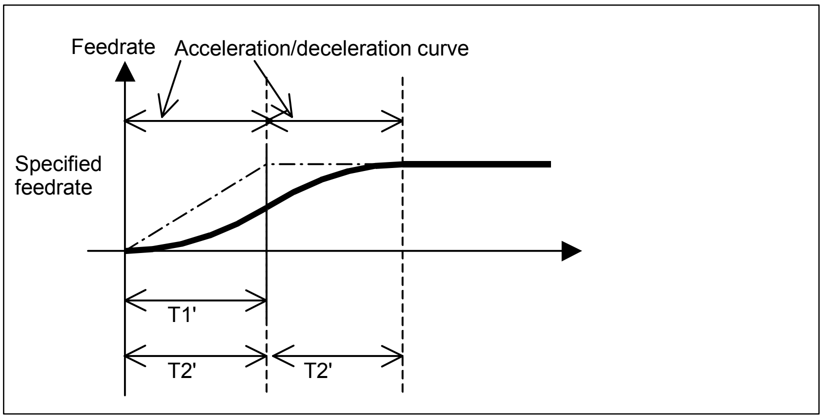
- Function for changing time constant of bell-shaped acceleration/deceleration

Bell-shaped acceleration/deceleration before interpolation is performed using the parameter-set acceleration and acceleration change time. If a low feedrate is specified, a linear acceleration/ deceleration may not achieve the specified acceleration as shown:



T1 : Time obtained from specified feedrate and specified acceleration (specified feedrate/acceleration (parameter No. 1660))
T2 : Acceleration change time (parameter No. 1772)

In such a case, set bit 3 (BCG) of parameter No. 7055 to 1. Then, the internal acceleration and vector time constant of acceleration/deceleration before interpolation are changed to make the acceleration/deceleration pattern as close as possible to the optimum bell-shaped acceleration/deceleration before interpolation based on a specified acceleration/deceleration reference speed, and so acceleration/deceleration time is reduced.



There are three methods for specifying the acceleration/deceleration reference speed.

(1) Specifying the speed using an F in a G05.1 Q1 block
(2) Setting the speed on parameter 7066
(3) Setting the speed specified with the F command issued at the start of cutting as the reference speed

When F is specified in a G05.1Q1 block, the specified feedrate is assumed to be the acceleration/deceleration reference speed. This command can be used only in the feed per minute mode.
If no F command is specified in a G05.1Q1 block, the feedrate specified in parameter No. 7066 is assumed to be the acceleration/deceleration reference speed. If 0 is set in parameter No. 7066, the F command specified in the cutting start block is assumed to be the acceleration/deceleration reference speed.

- Automatic feedrate control function

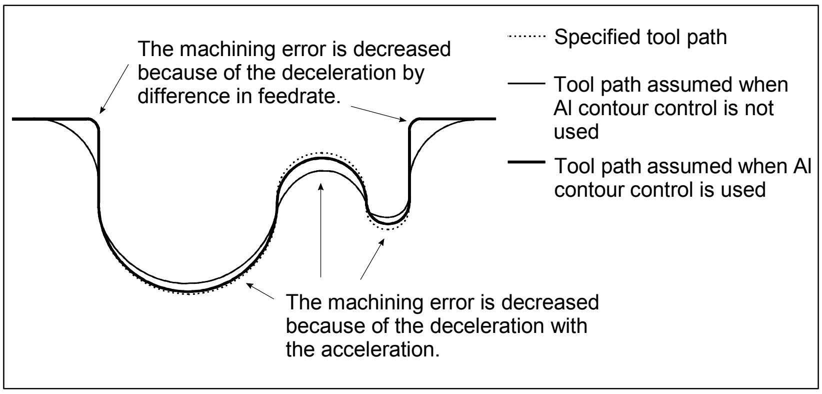
In AI contour control mode, the feedrate is automatically controlled by the reading-ahead of blocks.

The feedrate is determined using the following conditions. If the specified feedrate exceeds the determined feedrate, acceleration/ deceleration before interpolation is performed to achieve the determined feedrate.

<1> Feedrate changes on each axis at a corner and the permissible feedrate change that has been set

<2> Expected acceleration on each axis and the permissible acceleration that has been set

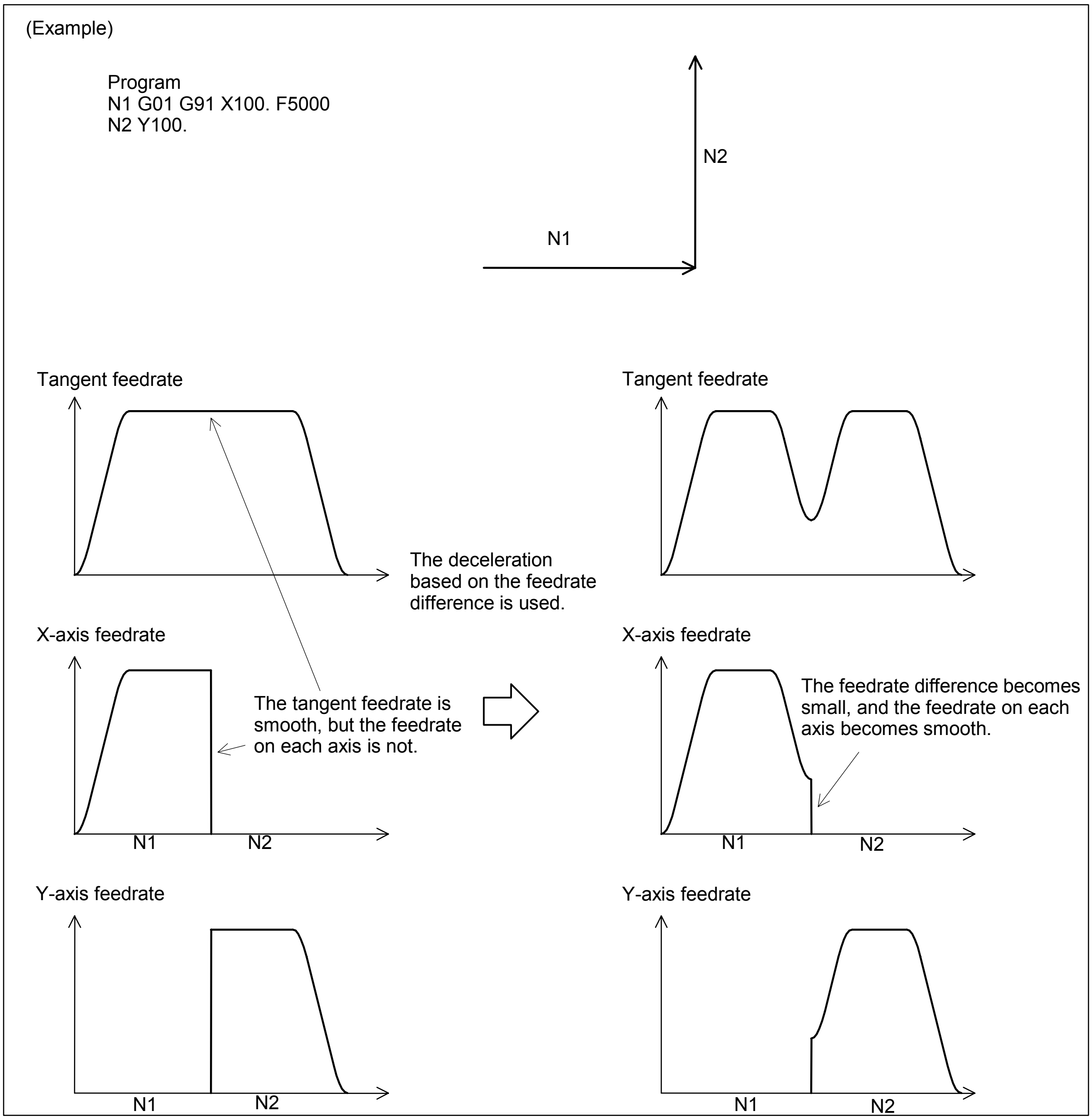
<3> Cutting load that is expected from the travel direction on the Z-axis



For details, see the explanation of each function.

- Speed control based on the feedrate difference on each axis at a corner

By using the speed control based on the feedrate difference on each axis at a corner, if a feedrate change occurs on an axis on each axis at a corner, the feedrate is determined so that any feedrate difference exceeding the permissible feedrate difference on that axis that has been set for parameter No. 1783 does not occur, and deceleration is automatically performed.

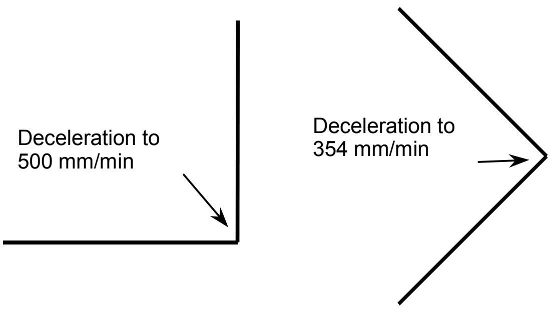


The method of deceleration based on the feedrate difference differs depending on the setting made for parameter FNW (bit 6 of No. 19500).

If "0" is set, the largest feedrate that does not exceed the permissible feedrate difference set for parameter No. 1783 is assumed to be the deceleration feedrate.

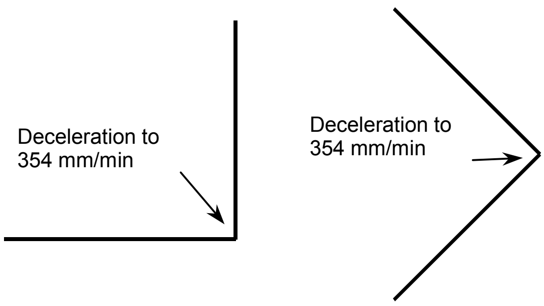
In this case, the deceleration feedrate differs if the travel direction differs, even if the shape is the same.

(Example)
If parameter FNW (bit 6 of No. 19500) = 0 and the
permissible feedrate difference = 500 mm/min (on all axes)



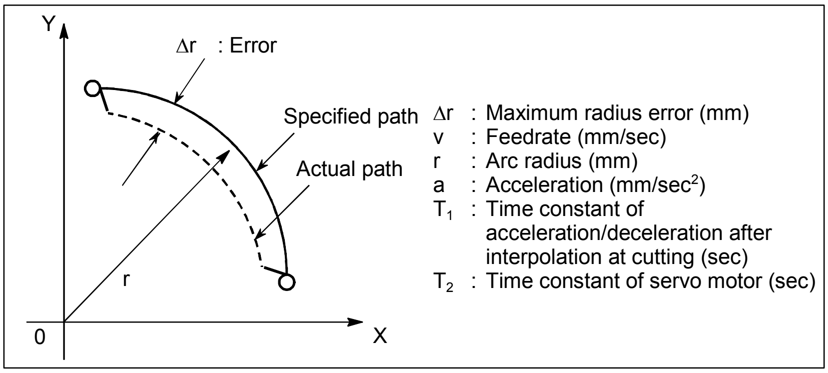
If "1" is set, the feedrate is determined not only with the condition that the permissible feedrate difference and permissible acceleration on each axis are not exceeded, but also that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If 1 is set for this parameter, the deceleration feedrate determined with the feedrate difference may be up to 30% lower than that determined if 0 is set.

(Example)
If parameter FNW (bit 6 of No. 19500) = 1 and
permissible feedrate difference = 500 mm/min (on all axes)



- Speed control with acceleration in circular interpolation

When high-speed cutting is performed in circular interpolation, helical interpolation, or spiral interpolation, the actual tool path has an error with respect to the programmed path. In circular interpolation, this error can be approximated from the equation given below.



$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2)\frac{v^2}{r} = \frac{1}{2}(T_1^2 + T_2^2)\cdot a \quad \text{.................................. (Equation 1)}$$

In actual machining, permissible error Δr is given, so the maximum permissible acceleration a (mm/sec^2) in equation 1 is determined.
When a specified feedrate causes the radial error from an arc having a programmed radius to exceed the permissible error, speed control with acceleration in circular interpolation automatically clamps the arc-cutting feedrate by using parameter settings.

Let the permissible acceleration calculated from the permissible acceleration set for each axis be A. Then, maximum permissible feedrate v with programmed radius r is expressed as follows:

$$v = \sqrt{A \cdot r} \quad \text{.. (Equation 2)}$$

If a specified feedrate exceeds feedrate v obtained from equation 2, the feedrate is clamped at feedrate v automatically.
The permissible acceleration is specified in parameter No. 1735. If there is a difference in permissible acceleration between two axes for circular interpolation, the lower acceleration is regarded as the permissible acceleration.
If the radius of an arc is small, too small value can be calculated as deceleration v. In such a case, the lower feedrate limit can be set in parameter No. 1732 to prevent the feedrate from being decreased too much.

- Speed control with the acceleration on each axis

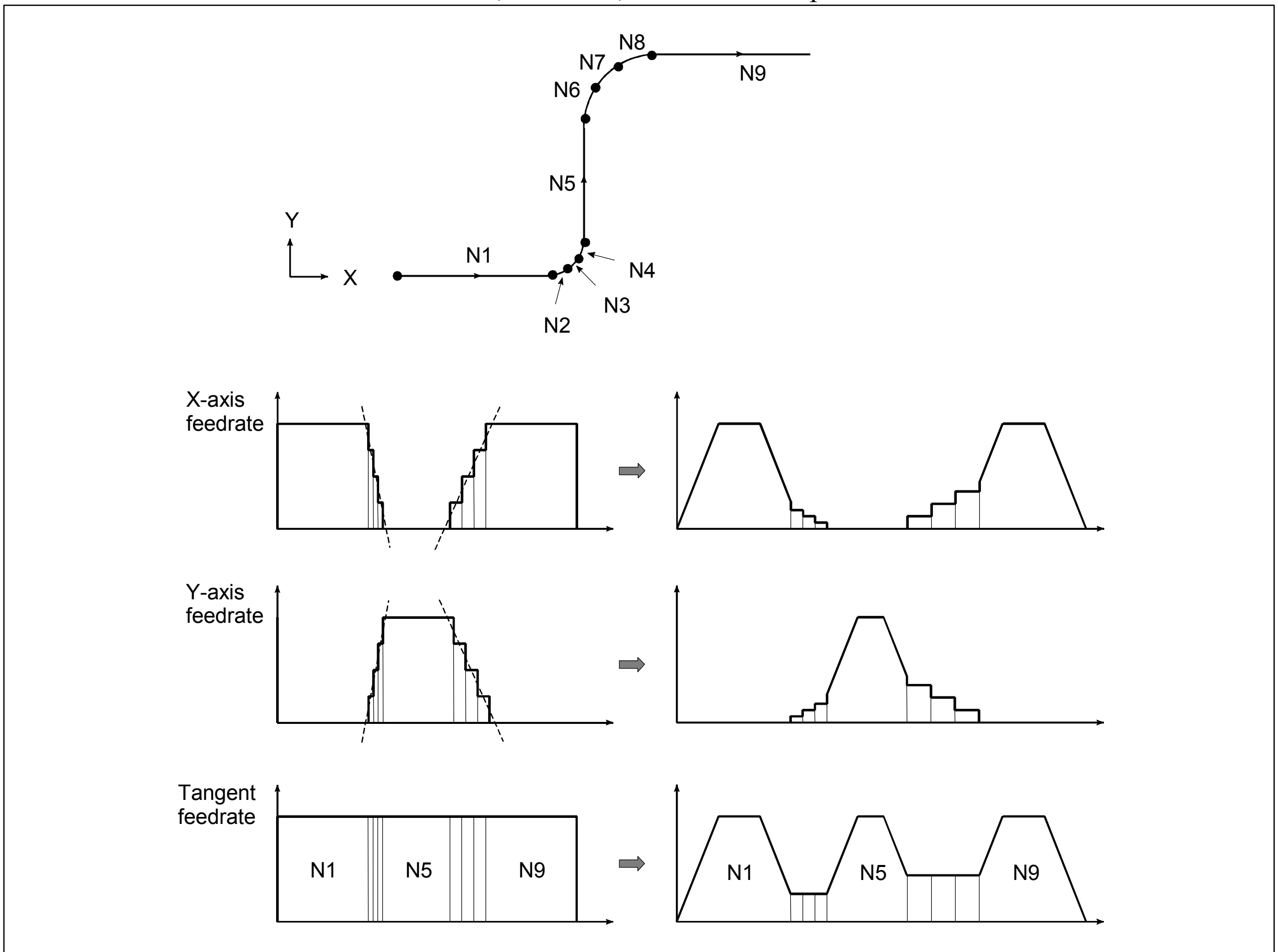
When consecutive small lines are used to form a curve, as in the example shown in the figure below, the feedrate differences on each axis at the individual corners are not very large. Thus, deceleration with the feedrate differences is not effective. Consecutive small feedrate differences, however, cause a large acceleration on each axis, as a whole.

In such a case, deceleration can be performed to reduce the impact on the machine and the machining error caused by too large an acceleration. The deceleration feedrate is determined to be the feedrate that does not cause the acceleration on each axis to exceed the permissible acceleration set for parameter No. 1737.

The deceleration feedrate is determined for each corner. The actual feedrate is the smaller of the deceleration feedrate determined at the start point of the block and that determined at the end point.

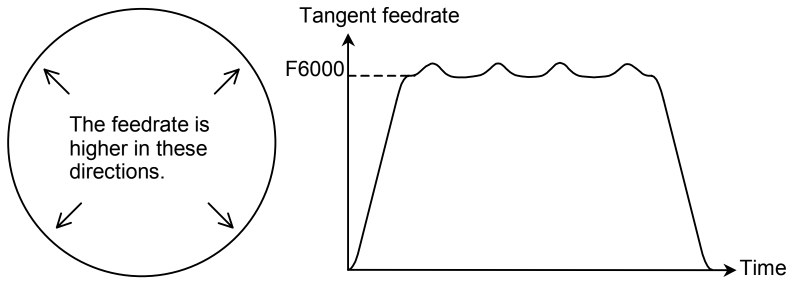
Depending on the specified figure, a very low deceleration feedrate may be calculated. In such a case, the lower feedrate limit can be set in parameter No. 1738 to prevent the feedrate from being decreased too much.

In the following example, the acceleration (gradient of the broken line in the feedrate graph) at too large at corners N2 to N4 and N6 to N8 and, therefore, deceleration is performed.



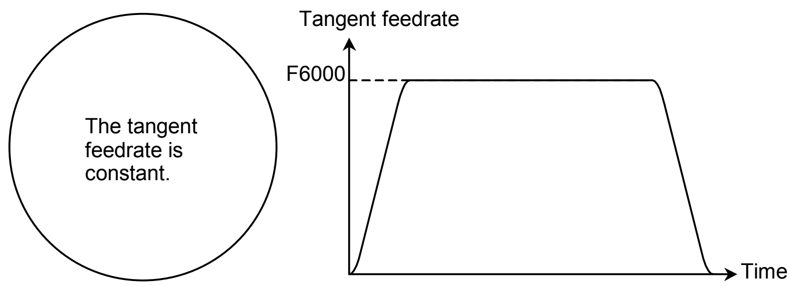
The method of determining the feedrate with the acceleration differs depending on the setting of parameter FNW (bit 6 of No. 19500).
If "0" is set, the highest feedrate that does not cause the permissible acceleration set for parameter No. 1737 to be exceeded is assumed to be the deceleration feedrate. In this case, the deceleration feedrate differs depending on the travel direction even if the shape is the same, as shown in the figure below.

(Example) If a circular shape with a radius of 10 mm is specified with small line blocks
Parameter FNW (bit 6 of No. 19500) = 0
Permissible acceleration = 1000 mm/s^2 (on all axes)



If "1" is set, the feedrate is determined with not only the condition that the permissible acceleration on each axis is not exceeded but also the condition that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If 1 is set for this parameter, the deceleration feedrate determined with the feedrate difference or acceleration may be up to 30% lower than that determined if 0 is set.

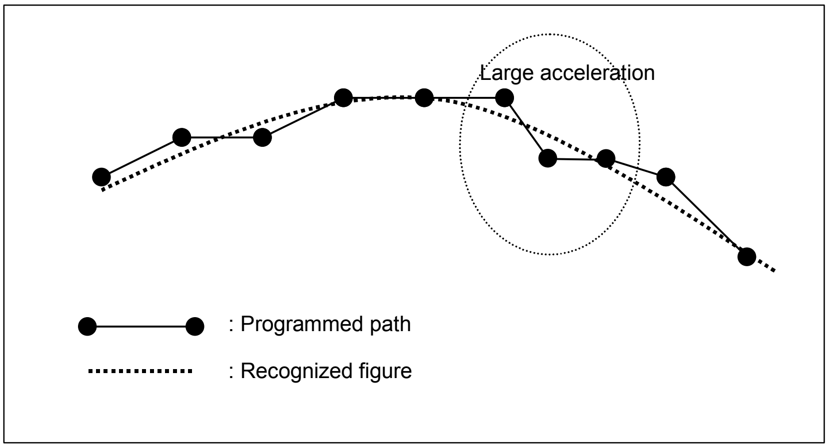
(Example) If a circular shape with a radius of 10 mm is specified with small line blocks
Parameter FNW (bit 6 of No. 19500) = 1,
radius = 10 mm, permissible acceleration = 1000 mm/s^2 (on all axes)



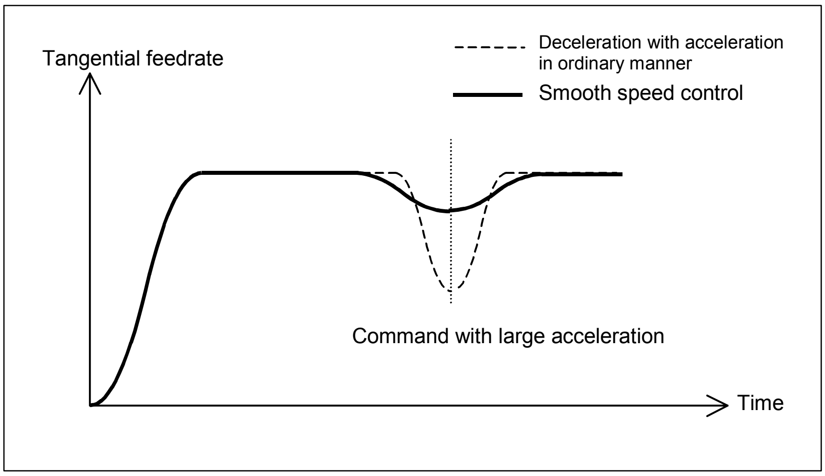
NOTE
In circular interpolation, the tangent feedrate is constant regardless of the setting of the parameter.

- Smooth speed control

In speed control with acceleration, the smooth speed control function recognizes the entire figure from preceding and following blocks including blocks read ahead to make a smooth feedrate determination. When a curve is specified with successive minute straight lines, programmed values are rounded to the least input increment before issued, so the machining profile is approximated with a broken line. When the feedrate is determined with acceleration in an ordinary manner, an optimum feedrate is automatically calculated exactly for a programmed figure, so a large acceleration may result depending on the command, which can lead to deceleration. In such a case, the use of smooth speed control enables speed control by recognizing the entire figure, which provides smooth speed control while suppressing local deceleration, therefore increasing the feedrate.



Also for a part of a programmed figure in which a large acceleration would be required, the acceleration is obtained based on the figure recognized from multiple blocks, and the feedrate is determined so that the acceleration is within the permissible acceleration set in parameter No. 1737.

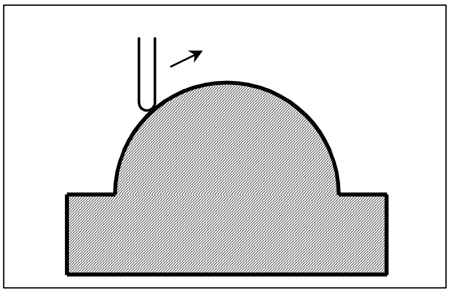


Smooth speed control obtains the acceleration by using the figure recognized from the preceding and following blocks including blocks read ahead, so smooth speed control is enabled even in parts in which the acceleration increases.
Smooth speed control is enabled under the following conditions:
<1> Speed control with acceleration is enabled in the AI contour control mode.
<2> Successive linear interpolation commands are specified.
<3> Bit 0 (HPF) of parameter No. 19503 is set to 1.

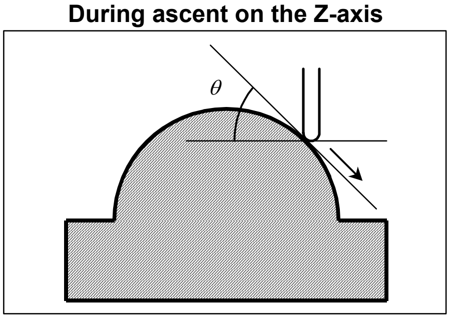
> ⚠ **CAUTION**
> When smooth speed control is used, the feedrate in a certain figure such as a corner may become larger than the feedrate obtained by ordinary speed control with acceleration. For corners, set parameter No. 1783, which is the permissible feedrate difference parameter for speed control with the feedrate difference at corners, to perform appropriate deceleration by speed control with the corner feedrate difference.

- Speed control with the cutting load

Usually, the cutting resistance produced when machining is performed with the bottom of the cutter as the tool lowers along the Z-axis is greater than the cutting resistance produced when machining is performed with the side of the cutter as the tool rises along the Z-axis. Therefore, deceleration is required.
In AI contour control, the tool travel direction on the Z-axis is used as a condition for calculating the machining feedrate.
This function is enabled when parameter ZAG (bit 4 of No. 8451) is set to 1.



During ascent on the Z-axis



During descent on the Z-axis

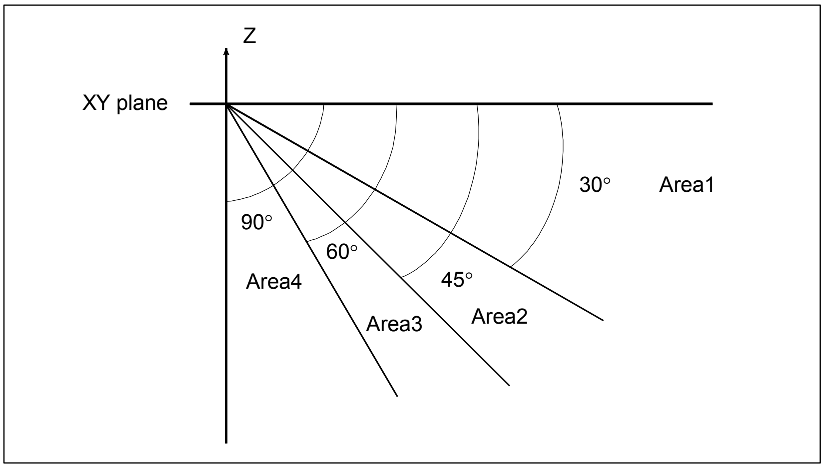
The descent angle θ during descent on the Z-axis (angle formed by the XY plane and the tool center path) is as shown in the figure. The descent angle is divided into four areas, and the override values for the individual areas are set for the following parameters:

Parameter No. 8456 for area 2
Parameter No. 8457 for area 3
Parameter No. 8458 for area 4

For area 1, however, no parameter is available, and an override of 100% is used at all times. The feedrate obtained according to other feedrate control is multiplied by the override value of the area to which descent angle θ belongs.

Area1	$0^\circ \le \theta < 30^\circ$
Area2	$30^\circ \le \theta < 45^\circ$
Area3	$45^\circ \le \theta < 60^\circ$
Area4	$60^\circ \le \theta < 90^\circ$

The feedrate can be overridden with an inclination by setting bit 1 (ZG2) of parameter No. 19515 to 1. In this case, specify the override value for area 1 in parameter No. 19516.



⚠ CAUTION

1 The speed control with the cutting feed is effective only when the tool is parallel with the Z-axis. Thus, it may not be possible to apply this function, depending on the structure of the machine used.
2 In the speed control with the cutting feed, the travel direction on the Z-axis is determined with the appropriate NC command. If, therefore, manual intervention is performed on the Z-axis with manual absolute on, or if a mirror image is applied on the Z-axis, the direction on the Z-axis cannot be determined. When using the speed control with the cutting load, do not use these functions.
3 When performing three-dimensional coordinate conversion, determine the descent angle on the Z-axis using the converted coordinate system.
4 Speed control with the cutting load is enabled for all interpolations in the AI contour control mode. This function, however, can be made valid only for linear interpolations by setting bit 4 (ZOL) of parameter No. 19503 to 1.

- Ignoring feedrate commands

In a block in which AI contour control is enabled, all feedrate commands (F commands) can be ignored by setting parameter NOF (bit 7 of No. 8451).
The term feedrate commands, as used here, refer to the following commands:

<1> Modal F commands before the block in which AI contour control is enabled
<2> F commands and modal F commands in the block in which AI contour control is enabled

When the feedrate commands are ignored, it is assumed that the upper feedrate limit specified for parameter No. 8465 is specified.
Note, however, that any issued F commands and modal F commands are stored within the CNC.
Thus, in a block in which AI contour control changes from the enabled state to the disabled state, the modal values of the F commands described in <1> and <2> described above are used as modal F commands, instead of the modal values of the F commands calculated by AI contour control.

- Another example of determining the feedrate

If a specified feedrate exceeds the upper feedrate limit of AI contour control (in parameter No. 8465), the feedrate is clamped at the upper feedrate. The upper feedrate limit is clamped at the maximum cutting feedrate (parameter No. 1432).

Limitation

- Conditions for temporarily canceling the AI contour control mode

If one of the commands listed below is issued in the AI contour control mode, the AI contour control mode is canceled temporarily.
If the system becomes ready for AI contour control after it is canceled, the AI contour control mode is restored automatically.

- Positioning (rapid traverse)
- Single direction positioning
- Spindle positioning
- Rigid tapping
- Hypothetical axis interpolation
- Threading (single type, combined type)
- Electric gear box
- When no move command is specified
- One-shot G code other than the following:
  Tool offset
  Cutter compensation vector retention
  Cutter compensation corner rounding
  Exact stop

- Functions that cannot be specified in the AI contour control mode

In the AI contour control mode, the functions listed below cannot be specified. Before specifying these functions, turn off the AI contour control mode; after the command ends, turn on the mode again.

- Threading
- Circular threading
- Variable lead threading

Threading, circular threading, and variable lead threading can be specified in the AI contour control mode by setting bit 1 (THA) of parameter No. 1611. However, the AI contour control mode is automatically canceled.

19.2 JERK CONTROL

19.2.1 Speed Control with Change of Acceleration on Each Axis

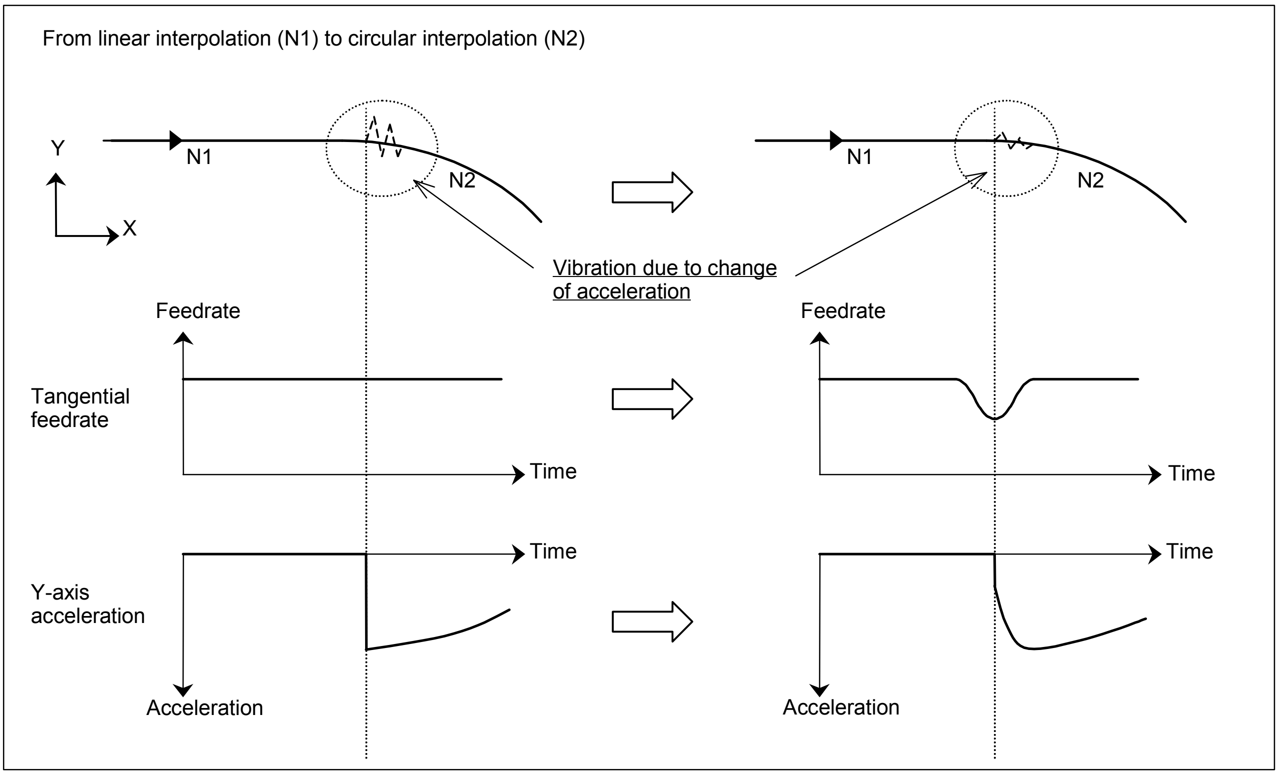
Overview

In portions in which acceleration changes largely, such as a portion where a programmed figure changes from a straight line to curve, vibration or shock on the machine may occur. Speed control with change of acceleration on each axis is a function to suppress machining errors due to vibration and machine shock generated by change of acceleration. This function obtains a feedrate so that change of acceleration is within the parameter-set permissible acceleration change amount for each axis, and performs deceleration by using acceleration/deceleration before interpolation.

> ⚠ **CAUTION**
> Before speed control with change of acceleration on each axis can be used, the options for jerk control and AI contour control are required.

Explanation

In the following example, the Y-axis acceleration changes largely at the contact point between a linear interpolation and circular interpolation, so deceleration is performed.



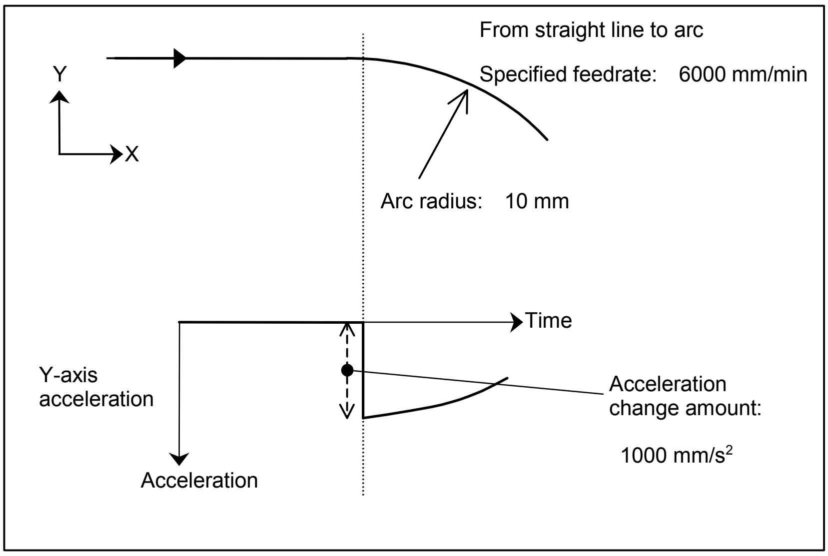
- Setting the permissible acceleration change amount

The permissible acceleration change amount for each axis is set in parameter No. 1788. When 0 is set in this parameter for a certain axis, speed control with change of acceleration is not performed for that axis.

- Parameter setting example

Suppose a figure shown below in which a straight line is followed by an arc. Let the specified feedrate and the arc radius be 6000 mm/min and 10 mm, respectively. Then, the Y-axis acceleration change amount at the contact point of the linear and arc portions is obtained as follows:

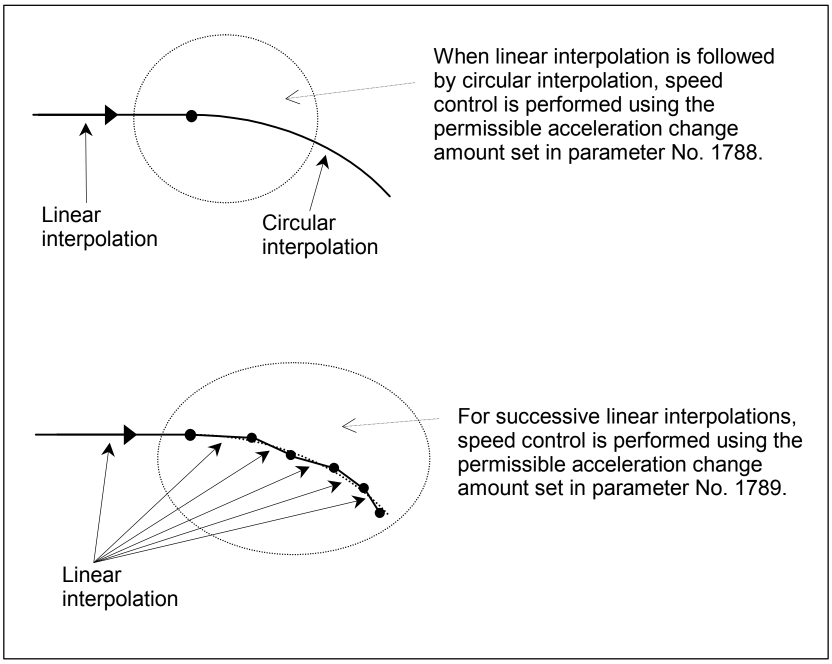
$$\frac{v^2}{r} = 1000mm/s^2$$



To suppress the change of acceleration to 300 mm/s^2, set 300 mm/s^2 for the Y-axis in parameter No. 1788.
Note that the change of acceleration is determined from the interpolation data of the CNC, so it may differ from the theoretical value.
The actual machine is affected by acceleration/deceleration and other factors, so the value to be set in the parameter should be determined after adjustments are made.

- For successive linear interpolations

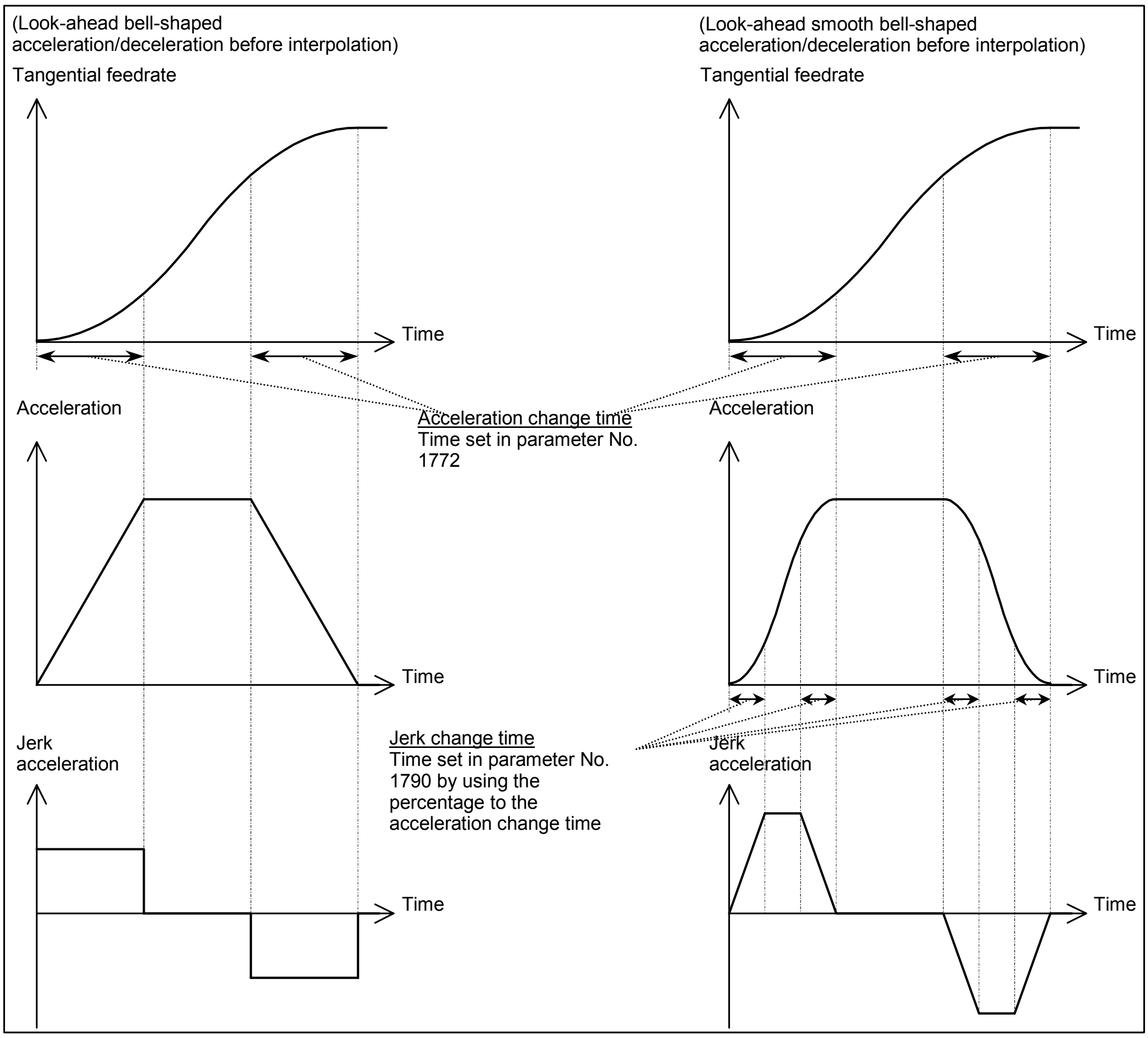
When there are successive linear interpolations, speed control with change of acceleration obtains the deceleration feedrate from the change in acceleration between the start point and end point of a specified block.
When a curve is specified using successive minute straight lines, programmed values are rounded to the least input increment before issued, so the machining profile is approximated with a broken line. The error due to rounding may increase change of acceleration, and especially when the line segments specified by blocks are short, deceleration is performed frequently. As a result, the machining speed cannot increase enough. In such a case, a relatively large value should be set in parameter No. 1789 as the permissible acceleration change amount for each axis in successive linear interpolations to improve the machining speed.
When a value other than 0 is set in parameter No. 1789 for an axis for which deceleration with change of acceleration is enabled, this setting is regarded as the permissible acceleration change amount at corners in which linear interpolations meet. (For portions where a linear interpolation and circular interpolation meet and where circular interpolations meet, the setting in parameter No. 1788 is used.)
When 0 is set in parameter No. 1789 for an axis, the setting in parameter No. 1788 specifying the ordinary permissible acceleration change amount is used even at a corner in which linear interpolations meet.
When smooth speed control is used in speed control with permissible acceleration in AI contour control (named temporarily), the deceleration feedrate is obtained from the change of acceleration calculated by smooth speed control.
Therefore, the deceleration feedrate may be higher than the ordinary deceleration feedrate.



19.2.2 Look-Ahead Smooth Bell-Shaped Acceleration/Deceleration before Interpolation

Overview

In look-ahead bell-shaped acceleration/deceleration before interpolation performs smooth acceleration/deceleration by changing the acceleration at a constant rate in specified acceleration change time.
In look-ahead smooth bell-shaped acceleration/deceleration before interpolation, the jerk change time is specified in parameter No. 1790 by using the percentage to the acceleration change time for look-ahead bell-shaped acceleration/deceleration before interpolation, and change of acceleration is also controlled so that the change is bell-shaped. This enables smoother acceleration/deceleration, therefore reducing machine vibration and shock due to acceleration/ deceleration.



> ⚠ **CAUTION**
> Before look-ahead smooth bell-shaped acceleration/deceleration before interpolation can be used, the option for jerk control and AI contour control II is required.

Explanation

- Setting the jerk change time

The jerk change time is set in parameter No. 1790 by using the percentage to the acceleration change time.
The actual jerk change time is represented by the percentage to the acceleration change time set in parameter No. 1772.
The jerk change time must be within a half of the acceleration change time, so the value to be set in the parameter ranges 0 to 50 (percent).
If 0 or a value beyond the specifiable range is specified in parameter No. 1790, look-ahead smooth bell-shaped acceleration/deceleration before interpolation is not enabled.

- Acceleration/deceleration before interpolation for linear type rapid traverse

When bell-shaped acceleration/deceleration is used in acceleration/deceleration before interpolation for linear type rapid traverse, enabling look-ahead smooth bell-shaped acceleration/deceleration before interpolation applies smooth bell-shaped acceleration/deceleration to acceleration/deceleration before interpolation for linear type rapid traverse.
In this case, the jerk change time is represented by the percentage set in parameter No. 1790 to the acceleration change time set in parameter No. 1672.

- Optimum torque acceleration/deceleration

When bell-shaped acceleration/deceleration is used in optimum torque acceleration/deceleration, enabling look-ahead smooth bell-shaped acceleration/deceleration before interpolation applies smooth bell-shaped acceleration/deceleration to optimum torque acceleration/deceleration.
In this case, the jerk change time is represented by the percentage set in parameter No. 1790 to the acceleration change time set in parameter No. 1672.

19.3 OPTIMUM TORQUE ACCELERATION/DECELERATION

Overview

This function enables acceleration/deceleration in accordance with the torque characteristics of the motor and the characteristics of the machines due to its friction and gravity and performs linear type positioning with optimum acceleration/deceleration during look-ahead acceleration/deceleration before interpolation.
Usually, because of the friction of the machine, gravity, the torque characteristics of the motor, and other factors, the acceleration/ deceleration performance (torque for acceleration/deceleration) is different with direction of movement, acceleration or deceleration. In this function, acceleration pattern of rapid traverse for the following situations, plus movement and acceleration, plus movement and deceleration, minus movement and acceleration, minus movement and deceleration can be set into parameters according to the torque for acceleration/deceleration of each situation.
Acceleration/deceleration can be performed according to these parameter setting, so that the most of the capability of the motor can be used and positioning time can be reduced.

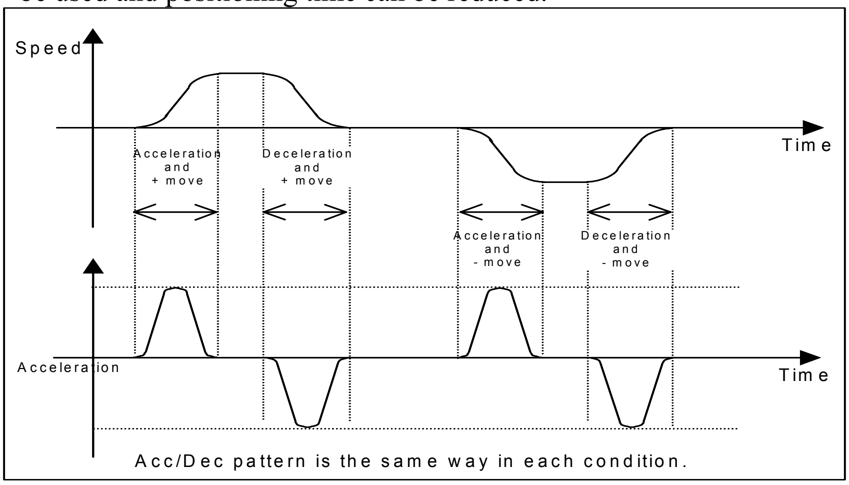


Fig. 19.3 (a) Conventional acceleration/declaration

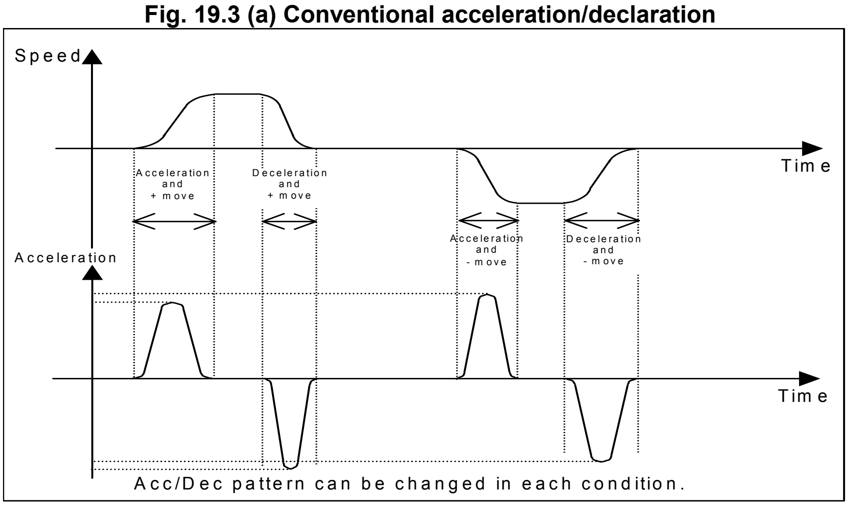


Fig. 19.3 (b) Acceleration/deceleration with this function

Explanation

Optimum torque acceleration/deceleration selects the acceleration pattern set with parameters on the basis of the axial movement direction and the acceleration/deceleration state, determines the acceleration for each axis from the current speed, and controls the tangential acceleration/deceleration for rapid traverse.

- Setting optimum torque acceleration/deceleration

When bit 0 (FAP) of parameter No. 19540 and bit 1 (LRP) of parameter No. 1401 are set to 1, and a value other than 0 is set for any one of the axes as the reference acceleration in parameter No. 1671 as shown below, rapid traverse is accelerated/decelerated by optimum torque acceleration/deceleration in the mode of look-ahead acceleration/deceleration before interpolation.

Table 19.3 (a) Optimum torque acceleration/deceleration

Parameter FAP (Optimum torque acc/dec)	Parameter LRP (Linear type positioning)	Reference acceleration	Bell-shaped acceleration change time	Acceleration pattern
1	1	Parameter (No.1671)	Parameter (No.1672)	See "Setting an acceleration pattern".

To enable bell-shaped acceleration/deceleration in addition to optimum torque acceleration/declaration, set the bell-shaped acceleration change time with parameter No.1672.

- Required conditions in addition to parameter setting

If the conditions for performing AI contour control and for acceleration/deceleration before interpolation are satisfied, optimum torque acceleration/deceleration is performed for accelerating and decelerating rapid traverse. If rapid traverse is subject to optimum torque acceleration/deceleration, after-interpolation acceleration/deceleration does not apply to rapid traverse.

- Cases in which optimum torque acceleration/ deceleration is disabled

If optimum torque acceleration/deceleration is disabled by parameter settings, or the required conditions other than parameter settings are not satisfied, acceleration/deceleration after interpolation is used to accelerate and decelerate rapid traverse.

- Setting acceleration pattern data

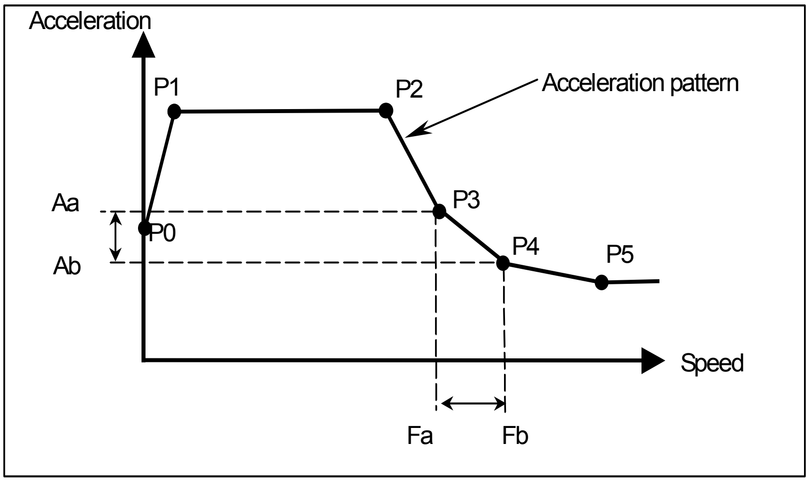


Fig. 19.3 (c) Setting acceleration pattern

Set the speed and the acceleration at each of the acceleration setting points P0 to P5 for each condition, plus movement and acceleration, plus movement and deceleration, minus movement and acceleration, minus movement and deceleration, and for each axis.
The line joining the acceleration setting points is regarded a acceleration pattern.
For example, while the speed is between Fa and Fb in the previous figure, the acceleration is calculated with Aa and Ab. Tangential acceleration is controlled not to exceed the calculated acceleration for each axis.

> ⚠ **CAUTION**
> It is not desirable to set an acceleration pattern in which a large acceleration is set right at a speed of 0, because this will shock the machine. For this reason, **be sure to set a low acceleration at a speed of 0**, as in the figure above.

Table 19.3 (b) Parameters for acceleration pattern

Acceleration setting point	Speed parameter	Acceleration parameter			
		During acceleration		During deceleration	
		During movement in plus direction	During movement in minus direction	During movement in plus direction	During movement in minus direction
P0	(Speed 0)	No.19545	No.19551	No.19557	No.19563
P1	No.19541	No.19546	No.19552	No.19558	No.19564
P2	No.19542	No.19547	No.19553	No.19559	No.19565
P3	No.19543	No.19548	No.19554	No.19560	No.19566
P4	No.19544	No.19549	No.19555	No.19561	No.19567
P5	No.1420	No.19550	No.19556	No.19562	No.19568

The speed at P0 is 0, and the speed at P5 is the rapid traverse rate specified with parameter (No. 1420). The speeds at P1 to P4 are to

be set into speed parameters Nos. 19541 to 19544 as ratio to the rapid traverse speed (parameter No. 1420).
Any acceleration setting point for which the speed parameter (Nos. 19541 to 19544) is set to 0 will be skipped, and the next point whose speed parameter is set to a non-zero value will be joined together as acceleration pattern.
The accelerations at P0 to P5 are to be set into acceleration parameters Nos. 19545 to 19568 as ratio to the reference acceleration. If any of the acceleration parameters Nos. 19545 to 19568 is set to 0, the acceleration is assumed 100% (Reference acceleration). Acceleration parameters should be set to 0 at the acceleration setting point whose speed parameter is set to 0.
If this function is enabled and parameter No.1671 for an axis are set to 0, the following values are assumed as the reference acceleration for that axis:
1000.0mm/sec^2, 100.0inch/sec^2, 100.0 deg/sec^2

- Example of setting acceleration pattern data

In this example, the machine is equipped with the α30/4000*i*s.
Motor speed at rapid traverse is 3000 (min^{-1}).

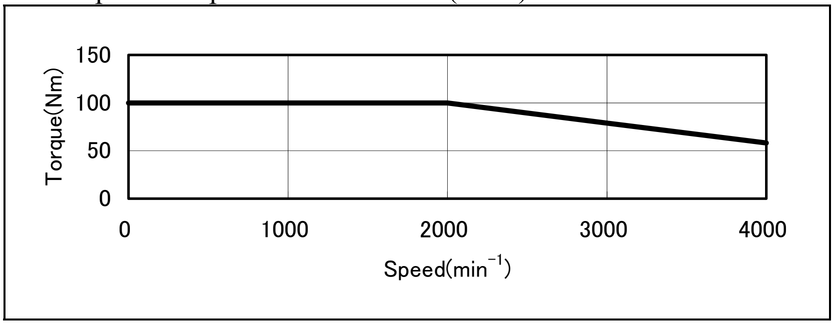


Fig. 19.3 (d) Speed-torque characteristics of model α30/4000*i*s

Specifications of the motor model α30/4000*i*s

Rotor inertia	:0.0099(Kgm^2)	
Maximum torque	:100(Nm)	Speed 0 to 2000(min^{-1})
Torque at rapid traverse	:79(Nm)	Speed 3000(min^{-1})
Minimum torque	:58(Nm)	Speed 4000(min^{-1})

It assumes that 10 (Nm) is needed for the friction torque, so that the torque for acceleration/deceleration is shown as the following figure.
Because the friction torque is different on each machine, it is necessary to observe the actual torque output on the machine.

Maximum torque	:90(=100-10)(Nm)	Speed 0 to 2000(min^{-1})
Torque at rapid traverse	:69(=79-10)(Nm)	Speed 3000(min^{-1})
Minimum torque	:48(=58-10)(Nm)	Speed 4000(min^{-1})

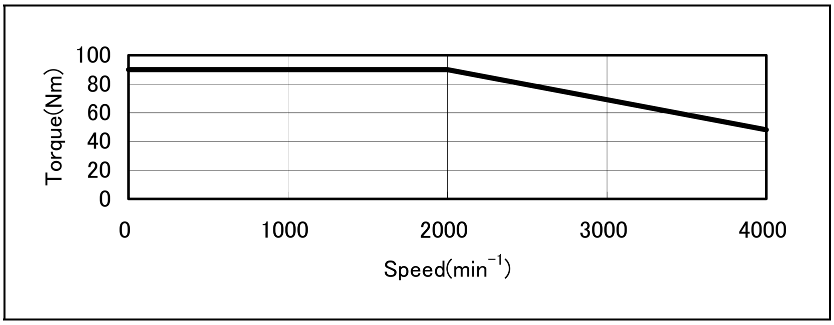


Fig. 19.3 (e) Torque for Acc/Dec with consideration of friction

Let the torque be x (Nm), the inertia be y (Kgm^2), and the ball screw pitch p (mm), then the acceleration A is calculated as follows:

$$A = \frac{x[N \cdot m]}{y[kg \cdot m^2]} \times \frac{p}{2\pi}[mm] = \frac{x([kg \cdot m/\sec^2][m])}{y[kg \cdot m^2]} \times \frac{p}{2\pi}[mm]$$

$$= \frac{x \times p}{2\pi \times y}[mm/\sec^2]$$

Machine specification is assumed as follows,

Ball screw pitch : 16 (mm)

Inertia : The machine inertia is to be 2.0 times higher than that of the rotor.
(Rotor inertia: 0.0099 (Kgm^2))

The acceleration at maximum torque 90 (Nm) is,

$$\frac{90 \times 16}{2\pi \times 3.0 \times 0.0099} = 7717[mm/\sec^2]$$

The acceleration at torque 69 (Nm) in rapid traverse 3000 (min^{-1}) is,

$$\frac{69 \times 16}{2\pi \times 3.0 \times 0.0099} = 5916[mm/\sec^2]$$

From the above data, the parameters related to acceleration pattern is given in the table below.
The example assumes that the acceleration is the same regardless of whether acceleration or deceleration is in progress or whether the movement is in the plus or minus direction.

- When setting the speed at P1, use the following value as a rough guide:
  If a bell-shaped acceleration change time (parameter No. 1672) is set, use as a rough guide the ratio of the following speed to the rapid traverse rate:
  Reference acceleration × T2/4
  where T2 is the bell-shaped acceleration change time (msec).
  For example, assume T2 to be 40 msec, then the speed at P1 should be approximately
  4124 [mm/sec^2]×40 [msec]/4
  =4124 [mm/sec^2]×60^2×40 [msec]/60000/4
  =2474 [mm/min]
- If no bell-shaped acceleration change time (parameter No. 1672) is set, use about 5% of the rapid traverse rate as a rough guide.

Table 19.3 (c) Example of setting parameters related to acceleration pattern

	Parameter No.	Setting	Unit	Remarks
Rapid traverse rate	1420	48000.	mm/ min	The ball screw pitch is assumed 16 mm, so that the rapid traverse rate is 48000 mm/min at the maximum speed 3000 (min^{-1}).
Reference acceleration	1671	4124.	msec	Reference acceleration is 4124mm/sec^2.
Speed at P1	19541	515	0.01%	- Assuming that parameter No. 1672 (bell-shaped acceleration change time) is set to 40 (msec), the speed at P1 will be 2474 mm/min from the calculation described on the previous page. Set its ratio to the rapid traverse rate, or 5.15%. 0.0515=2474/48000 - If parameter No. 1672 (bell-shaped acceleration change time) is not set, set about 5.00%.
Speed at P2	19542	6666	0.01%	Since the torque characteristic is constant (90 Nm) in speeds reaching 2000 min^{-1}, set P2 to the ratio of the speed (32000 mm/min) at 2000 min^{-1} to the rapid traverse rate (48000 mm/min). 0.6666=32000 / 48000
Speeds at P3 to P4	19543 to 19544	0	0.01%	P3 to P4 are skipped because the torque drops almost linearly from the speed 2000 (min^{-1}) to 3000 (min^{-1}).
Acceleration at P0	19545,19551, 19557,19563	9356	0.01%	Set half the acceleration at P1, or 9356.
Acceleration at P1	19546,19552, 19558,19564	18712	0.01%	At P1, 90(Nm) can be used for the acceleration/deceleration, so set the ratio 7717 (mm/sec^2) to 4124 (mm/sec^2). (1.8712 = 7717/4124)
Acceleration at P2	19547,19553 19559,19565	18712	0.01%	At P2, set the same speed as that at P1.
Acceleration at P3 to P4	19548 to 19549, 19554 to 19555, 19560 to 19561 19566 to 19567	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19550,19556 19562,19568	14345	0.01%	At P5, 69(Nm) can be used for the acceleration/deceleration, so set the ratio 5916 (mm/sec^2) to 4124 (mm/sec^2). (1.4345 = 5916 / 4124)

With the above parameter settings, the acceleration pattern will be shown as the following figure. From speeds from 0 mm/min to 2474 mm/min, the acceleration as calculated in accordance with the acceleration pattern is applied; from speeds from 2474 mm/min to 32000 mm/min, an acceleration of 7716 mm/sec^2; and from speeds

from 32000 mm/min to 48000 mm/min, the acceleration as calculated in accordance with the acceleration pattern.

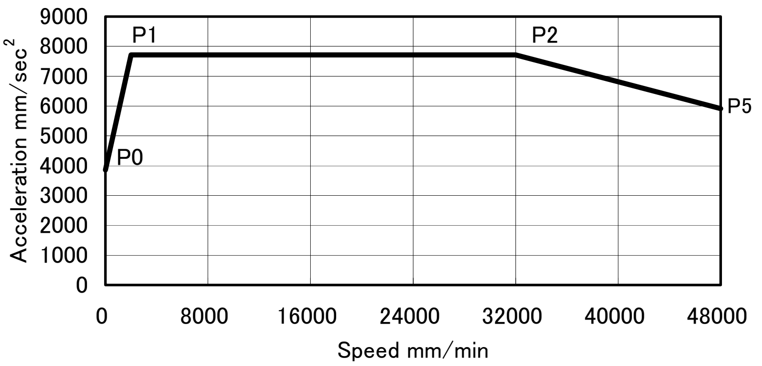


Fig. 19.3 (f) Acceleration pattern with consideration of friction

> **NOTE**
> The values in the model α30/4000*i*s speed-torque characteristic diagram are just typical ones. The values will change depending on the digital servo software, parameters, input voltage, and other factors.
> The optimum acceleration will, therefore, change due to the characteristics of the machine.

- Examples of setting if the acceleration pattern differs depending on whether acceleration or deceleration is in progress and whether the movement is in the minus or plus direction

From the effect of gravity and friction, torque for acceleration/deceleration is different on each condition, such as acceleration, deceleration or plus move (up), minus move (down).
The following example is for the vertical axis and gravity and friction torque are assumed as follows.
Torque of gravity : 20 (Nm)
Torque of friction : 10 (Nm)
Because these value are different on each machine, it is necessary to observe the output torque on the actual machine for deciding acceleration pattern. The conditions are the same as the previous example.
(Condition)

Motor speed at rapid traverse : 3000(min^{-1})
Ball screw pitch : 16(mm)
Inertia : The machine inertia is to be 2.0 times higher than that of the rotor.
Rotor inertia : 0.0099(Kgm^2)
Maximum torque : 100(Nm) Speed 0 to 2000(min^{-1})
Torque at rapid traverse : 79(Nm) Speed 3000(min^{-1})
Minimum torque : 58(Nm) Speed 4000(min^{-1})

(1) In case of plus move (up) and acceleration
Because torque of Gravity and friction work against the output torque of motor, the torque for acceleration/deceleration is as follows.

Maximum torque	: 70(=100-20-10) (Nm) Speed 0 to 2000(min^{-1})
Torque at rapid traverse	: 49(=79-20-10) (Nm) Speed 3000(min^{-1})
Minimum torque	: 28(=58-20-10) (Nm) Speed 4000(min^{-1})

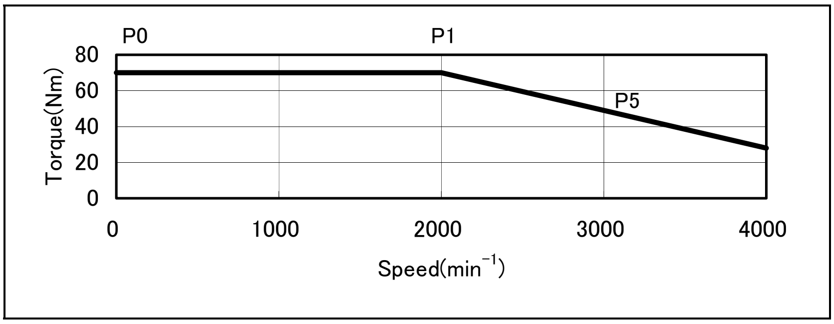


Fig. 19.3 (g) Torque for Acc/Dec in case of + move and acceleration

Parameter setting is as follows,

	Parameter No.	Setting	Unit	Remarks
Acceleration at P0	19545	7277	0.01%	Set half the acceleration at P1, or 7277.
Acceleration at P1-P2	19546,19547	14554	0.01%	At P1 and P2, 70(Nm) can be used for the acceleration/ deceleration, so set the ratio 6002 (mm/sec^2) to 4124 (mm/sec^2). (1.4554 = 6002 / 4124)
Acceleration at P3-P4	19548 to 19549	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19550	10187	0.01%	At P5, 49(Nm) can be used for the acceleration/deceleration, so set the ratio 4201 (mm/sec^2) to 4124 (mm/sec^2). (1.0187 = 4201 / 4124)

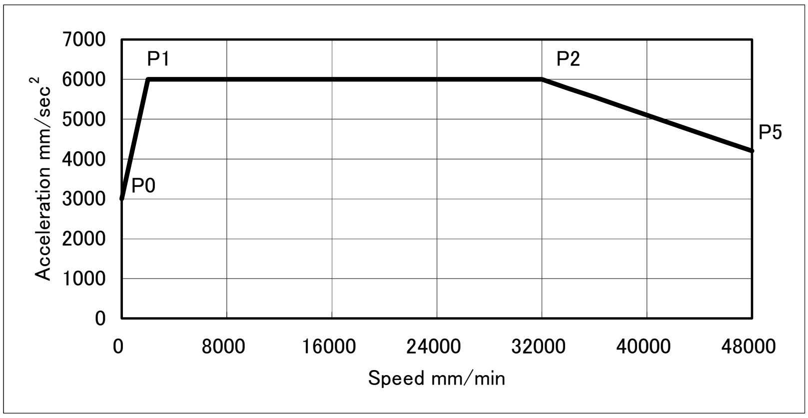


Fig. 19.3 (h) Acceleration pattern in case of + move and acceleration

(2) In case of plus move (up) and deceleration
Because torque of Gravity and friction work forward to the output torque of motor, the torque for acceleration/deceleration is as follows.

Maximum torque	: 130(=100+20+10) (Nm) Speed 0 to 2000(min^{-1})
Torque at rapid traverse	: 109(=79+20+10) (Nm) Speed 3000(min^{-1})
Minimum torque	: 88(=58+20+10) (Nm) Speed 4000(min-1)

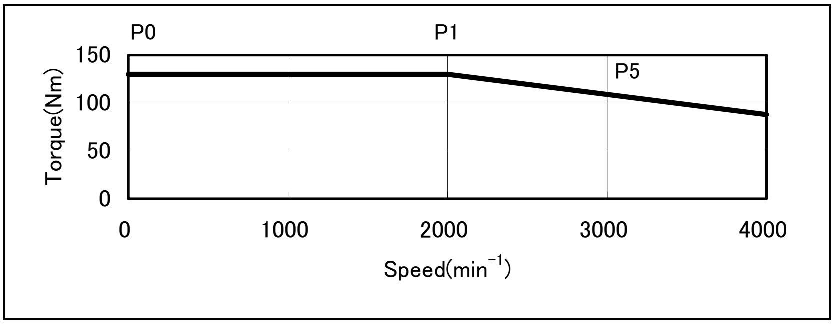


Fig. 19.3 (i) Torque for Acc/Dec in case of + move and deceleration

Parameter setting is as follows,

	Parameter No.	Setting	Unit	Remarks
Acceleration at P0	19557	13514	0.01%	Set half the acceleration at P1, or 13514.
Acceleration at P1-P2	19558, 19559	27027	0.01%	At P1 and P2, 130(Nm) can be used for the acceleration/ deceleration, so set the ratio 11146 (mm/sec^2) to 4124 (mm/sec^2). (2.7027 = 11146 / 4124)
Acceleration at P3-P4	19560 to 19561	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19562	22662	0.01%	At P5, 109(Nm) can be used for the acceleration/deceleration, so set the ratio 9346 (mm/sec2) to 4124 (mm/sec2). (2.2662 = 9346 / 4124)

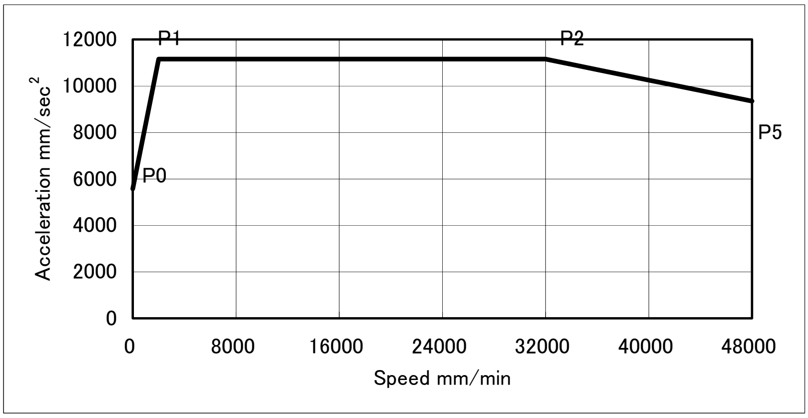


Fig. 19.3 (j) Acceleration pattern in case of + move and deceleration

(3) In case of minus move (down) and acceleration
Because torque of Gravity works forward to the output torque of motor and torque of friction works against the output torque of motor, torque for acceleration/deceleration is as follows.

Maximum torque	: 110(=100+20-10) (Nm) Speed 0 to 2000(min^{-1})
Torque at rapid traverse	: 89(=79+20-10) (Nm) Speed 3000(min^{-1})
Minimum torque	: 68(=58+20-10) (Nm) Speed 4000(min^{-1})

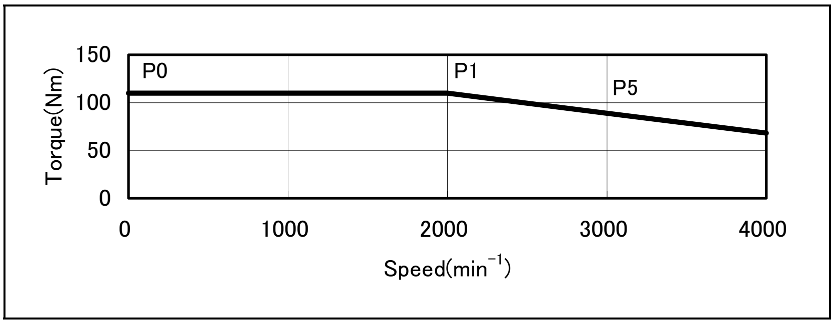


Fig. 19.3 (k) Torque for Acc/Dec in case of - move and acceleration

Parameter setting is as follows,

	Parameter No.	Setting	Unit	Remarks
Acceleration at P0	19551	11435	0.01%	Set half the acceleration at P1, or 11435.
Acceleration at P1-P2	19552,19553	22869	0.01%	At P1 and P2, 110(Nm) can be used for the acceleration/ deceleration, so set the ratio 9431 (mm/sec^2) to 4124 (mm/sec^2). (2.2869 = 9431 / 4124)
Acceleration at P3-P4	19554 to 19555	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19556	18504	0.01%	At P5, 89(Nm) can be used for the acceleration/deceleration, so set the ratio 7631 (mm/sec^2) to 4124 (mm/sec^2). (1.8504 = 7631 / 4124)

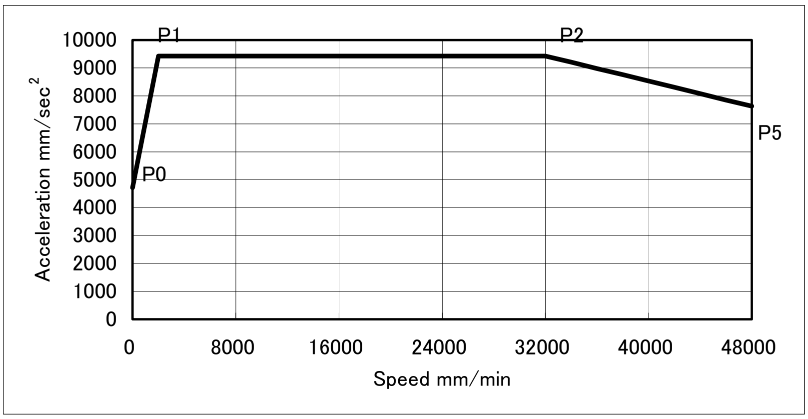


Fig. 19.3 (l) Acceleration pattern in case of - move and acceleration

(4) In case of minus move (down) and deceleration
Because torque of Gravity works against the output torque of motor and torque of friction works forward to the output torque of motor, torque for acceleration/deceleration is as follows.

Maximum torque	: 90(=100-20+10) (Nm) Speed 0 to 2000(min^{-1})
Torque at rapid traverse	: 69(=79-20+10) (Nm) Speed 3000(min^{-1})
Minimum torque	: 48(=58-20+10) (Nm) Speed 4000(min^{-1})

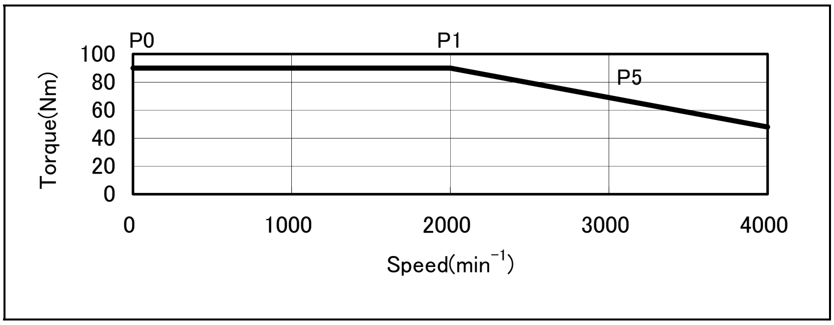


Fig. 19.3 (m) Torque for Acc/Dec in case of - move and deceleration

Parameter setting is as follows,

	Parameter No.	Setting	Unit	Remarks
Acceleration at P0	19563	9356	0.01%	Set half the acceleration at P1, or 9356.
Acceleration at P1-P2	19564,19565	18712	0.01%	At P1 and P2, 90(Nm) can be used for the acceleration/ deceleration, so set the ratio 7717 (mm/sec^2) to 4124 (mm/sec^2). (1.8712 = 7717 / 4124)
Acceleration at P3-P4	19566 to 19567	0	0.01%	0 is set because P3 to P4 are skipped.
Acceleration at P5	19568	14345	0.01%	At P5, 69(Nm) can be used for the acceleration/deceleration, so set the ratio 5916 (mm/sec^2) to 4124 (mm/sec^2). (1.4345 = 5916 / 4124)

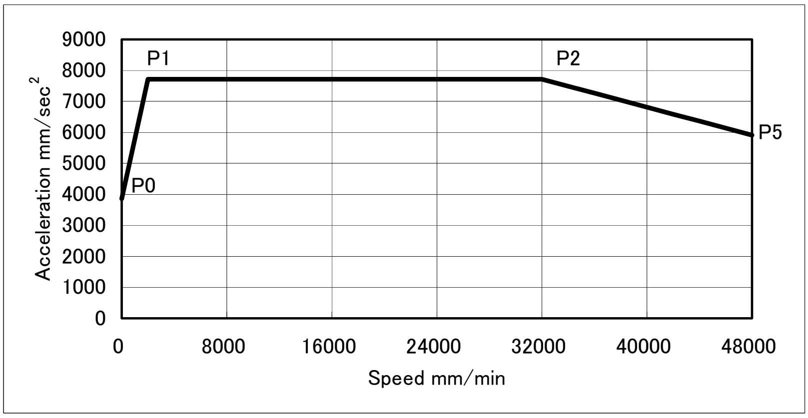


Fig. 19.3 (n) Acceleration pattern in case of - move and deceleration

Limitation

- Linear type positioning

Optimum torque acceleration/deceleration is not enabled unless linear-type positioning is set (bit 1 (LRP) of parameter No. 1401 is set to 1).

- Modes and conditions

Optimum torque acceleration/deceleration is enabled when look-ahead acceleration/deceleration before interpolation (or AI contour) control mode is active and the conditions under which look-ahead acceleration/deceleration before interpolation is performed are satisfied.

- Target axes

Optimum torque acceleration/deceleration cannot be performed for a specific axis only. All axes operated by programmed commands are targeted for optimum torque acceleration/deceleration. This means that the PMC axes are excluded.

- Acceleration pattern

In the same direction move, it is necessary that the acceleration during deceleration is set 1/3 of one during acceleration at least.

Also, the acceleration pattern data must be set so that the time required for the deceleration from the rapid traverse rate to a speed of 0 does not exceed 4000 (msec). This does not include the acceleration change time of bell-shaped acceleration/deceleration.

If the deceleration acceleration ratio or the time required for the deceleration to a speed of 0 exceeds the above range, alarm (DS1710) will be issued at the time rapid traverse is executed.

A slight error will occur between the specified acceleration and the actual acceleration.

- Relationship with the customer board

Optimum torque acceleration/deceleration cannot be used from the customer board.

- Tool center point control

Optimum torque acceleration/deceleration is disabled in the tool center point control mode (except startup and cancellation). In this case, positioning is accelerated/decelerated with the reference acceleration.

20 AXIS CONTROL FUNCTIONS

20.1 AXIS SYNCHRONOUS CONTROL

Overview

When a movement is made along one axis by using two servo motors as in the case of a large gantry machine, a command for one axis can drive the two motors by synchronizing one motor with the other. Moreover, by using a feedback signal from each motor, a positional difference (synchronous error) between the two motors is detected to compensate for the synchronous error. When a synchronous error exceeding a set value occurs, a synchronous error check can be made to issue an alarm and stop a movement along the axis.

An axis used as the reference for axis synchronous control is referred to as a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is referred to as a slave axis (S-axis).

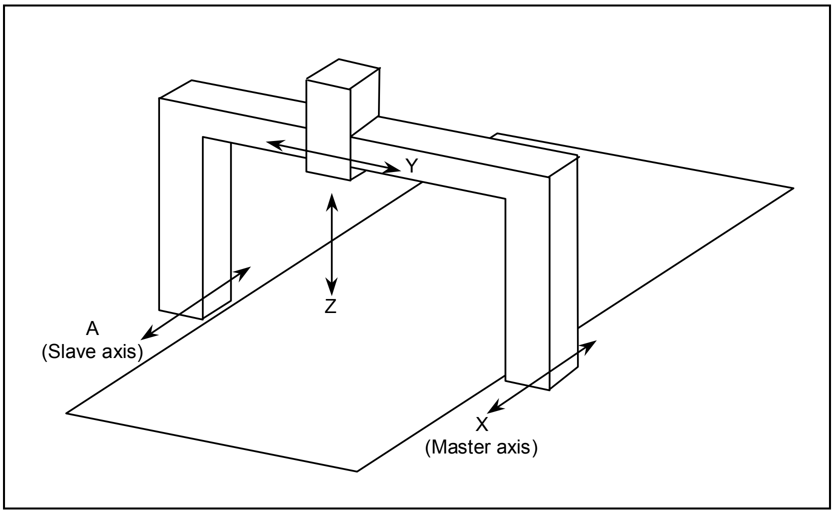


Fig. 20.1 (a) Example of machine with X and A being synchronous axes

Even when synchronous error compensation is not used, the synchronous establishment function can be used for automatic compensation to eliminate a machine coordinate error in cases such as emergency stop cancellation.

An external signal can be used to turn synchronization on and off. When synchronization is turned on and off using an external signal, synchronous error compensation cannot be used.

20.1.1 Axis Configuration for Axis Synchronous Control

Explanation

- Master axis and slave axis for axis synchronous control

An axis used as the reference for axis synchronous control is referred to as a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is referred to as a slave axis (S-axis).
By setting the axis number of a master axis in the parameter (No. 8311) of the slave axis, the axis configuration for axis synchronous control is determined.

- Synchronous operation and normal operation

Operation where axis synchronous control is turned on (enabled) to make a movement along the slave axis in synchronism with the master axis is referred to as synchronous operation. Operation where axis synchronous control is turned off (disabled) to make movements along the master axis and slave axis independently of each other is referred to as normal operation.

(Example)
Automatic operation when the master axis is the X-axis and the slave axis is the A-axis

In synchronous operation, movements are made along the X-axis and A-axis according to the programmed command Xxxxx for the master axis.
In normal operation, movements are made along the master axis and slave axis independently of each other as in the case of normal CNC control. The programmed command Xxxxx makes a movement along the X-axis. The programmed command Aaaaa makes a movement along the A-axis. The programmed command Xxxxx Aaaaa makes movements along the X-axis and A-axis at the same time.

The mode of operation can be switched between synchronous operation and normal operation by an input signal, or synchronous operation can be performed at all times. Which mode to use can be set using bit 5 (SCA) of parameter No. 8304.

- Switching between synchronous operation and normal operation by using an input signal

When bit 5 (SCA) of parameter No. 8304 is set to 0 for the slave axis, the signal SYNCx/SYNCJx (with x representing a slave axis number) is used to switch between synchronous operation and normal operation. When SYNCx/SYNCJx = 1, synchronous operation is selected. When SYNCx/SYNCJx = 0, normal operation is selected.
Synchronous error compensation cannot be used when the mode of operation is switched between synchronous operation and normal operation.

- Setting for using synchronous operation at all times

When bit 5 (SCA) of parameter No. 8304 for the slave axis is set to 1, synchronous operation is performed at all times, regardless of the setting of the signal SYNCx/SYNCJx.

- Synchronous control axis name

The name of a master axis and the name of a slave axis may be the same or may be different from each other.

- Restrictions on using the same name for the master axis and slave axis

If the same axis name is assigned to the master axis and slave axis, manual operation only is allowed in normal operation. Automatic operation and manual numeric command cannot be performed.

- Setting of an axis name subscript

A subscript can be attached to an axis name like X1, X2, XM, and XS. If the same axis name is used for multiple axes, and a unique subscript is assigned to each of those axes, the axes can be distinguished from each other on the screen display, or which of those axes issued an alarm can be identified.
Set a subscript in parameter No. 3131.

- Setting of multiple slave axes

One master axis can have multiple slave axes.

(Example)

In the example below, movements along the X1-axis and X2-axis are made in synchronism with the XM-axis.

Axis name indication	Controlled axis number	Axis name Parameter (No. 1020)	Subscript Parameter (No.3131)	Master axis number Parameter (No.8311)	Operation
XM	1	88	77	0	
Y	2	89	0	0	
X1	3	88	49	1	A movement is made in synchronism with the XM-axis.
X2	4	88	50	1	A movement is made in synchronism with the XM-axis.

When one master axis has multiple slave axes, synchronous error compensation, synchronous establishment, and synchronous error check are performed for each slave axis independently.

- Combination with tandem control

Tandem control can be used with each of the master and slave axes. The same restriction on axis arrangement as imposed in the case of normal tandem control is imposed. No particular restriction is imposed on axis synchronous control.

- Axis selection on the screen display

On a screen such as the current position display screen, a slave axis is also displayed. The display of a slave axis can be disabled by setting bit 0 (NDP) of parameter No. 3115 to 1 and setting bit 1 (NDA) of parameter No. 3115 to 1.

- Axis selection in actual cutting feedrate display

By setting bit 2 (SAF) of parameter No. 8303 to 1 for a slave axis, the slave axis can be included in an actual cutting feedrate display calculation during synchronous operation.

- Axis synchronous control with an absolute-position detector

When bit 7 (SMA) of parameter No. 8302 is set to 1 to attach an absolute-position detector, and bit 4 (APZ) of parameter No. 1815 for an axis placed in synchronous operation is turned off, APZ for the axis (axes) placed together in synchronous operation is also turned off.

- Slave axis mirror image

By setting parameter No. 8312, a mirror image can be applied to a slave axis placed in synchronous operation. When the mirror image function is enabled, the direction in which the absolute and relative coordinates change is the same as for the machine coordinates.
At this time, synchronization error compensation, synchronization establishment, synchronization error check, and correction mode cannot be used.
The mirror image set by bit 0 (MIR) of parameter No. 0012 cannot be applied to the slave axis. Because this mirror image differs from the mirror image set by parameter MIR, it does not affect input signal MIx (G106) or output signal MMIx (F108).

- External machine coordinate system shift

Bit 7 (SYE) of parameter No. 8304 can be set to 1 for the slave axis to shift the slave axis by the same amount as specified for the master axis when external machine coordinate system shift is specified by external data input/output for the master axis in synchronous control.

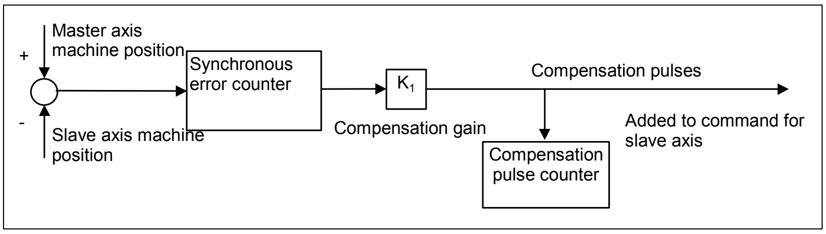
20.1.2 Synchronous Error Compensation

Explanation

When a synchronous error value exceeding the zero width set in parameter No. 8333 is detected, compensation pulses for synchronous error reduction are calculated and added onto the command pulses output for the slave axis. This compensation is not performed in servo-off state, servo alarm state, follow-up operation, and modification mode.
Compensation pulses are calculated by multiplying the synchronous error value between master and slave axes by a compensation gain.

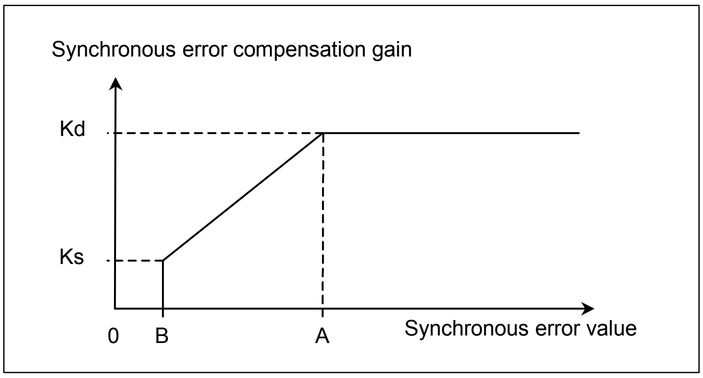
Compensation pulses = synchronous error × (Ci/1024)
Ci: Compensation gain (parameter No. 8334)

Before synchronous error compensation can be performed, bit 5 (SCA) of parameter No. 8304 must be set to 1 to perform synchronous operation at all times. Furthermore, the first synchronous establishment after power-up must already be performed by a manual reference position return or another operation.



- Synchronous error compensation smooth suppress function

When bit 6 (SMS) of parameter No. 8304 is set to 1, the synchronous error compensation smooth suppress function is enabled. With this function, another set of parameters for a synchronous error zero width and synchronous error compensation gain (B and Ks in the figure below) can be set. So, even a small synchronous error can be reduced smoothly as shown below.



A: Synchronous error zero width (parameter No. 8333)
B: Synchronous error zero width 2 (parameter No. 8335)
(0 < B < A)
Kd: Synchronous error compensation gain (parameter No. 8334)

Ks: Synchronous error compensation gain 2 (parameter No. 8336) ($0 < Ks < Kd$)
Er: Synchronous error value between the current master axis and slave axis
K: Current synchronous error compensation gain for Er

1. When $Er < B$, compensation is not performed. ($K = 0$)
2. When $B < Er < A$

   $$K = Ks + \frac{(Er - B)(Kd - Ks)}{A - B}$$

   Compensation is performed with the following gain:
3. When $Er > A$, compensation is performed with a gain of $K = Kd$.

20.1.3 Synchronous Establishment

Explanation

Upon power-up or after emergency stop cancellation, the machine positions on the master axis and slave axis under axis synchronous control are not always the same. In such a case, the synchronous establishment function matches the machine position on the master with that on the slave axis.

- Synchronous establishment method

The method of synchronous establishment used when synchronous error compensation is performed differs from that used when synchronous error is not performed.

- Synchronous establishment based on synchronous error compensation

When synchronous error compensation is performed (when bit 3 (CLP) of parameter No. 8304 for the slave axis is set to 1), synchronous establishment is performed in the same way as synchronous error compensation. This means that the positional difference between the master axis and slave axis is regarded as a synchronous error, and pulses produced by multiplying the positional difference by a synchronous error compensation gain are output for the slave axis. So, the parameter (No. 8334) for specifying a synchronous error compensation gain must be set before synchronous establishment can be performed.

If the parameter (No. 8333) for specifying a synchronous error compensation zero width is set, no further synchronous establishment is performed after the positional difference between the master axis and slave axis becomes the zero width or below.

- Synchronous establishment based on machine coordinates

To perform synchronous establishment when synchronous error compensation is disabled (when bit 3 (CLP) of parameter No. 8304 for the slave axis is set to 0), enable synchronous establishment based on machine coordinates by setting bit 7 (SOF) of parameter No. 8303 to 1.

This method of synchronous establishment outputs the machine coordinate difference between the master axis and slave axis as command pulses for the slave axis to establish synchronization. A machine coordinate difference is output at a time as command pulses. So, if the compensation value is large, the machine abruptly makes a large movement. Taking this into consideration, set a maximum allowable compensation value to be used for synchronous establishment in parameter No. 8325. As a maximum allowable compensation value, set a maximum allowable value by which the machine may move abruptly. If a compensation value is larger than the value set in this parameter, an alarm (SV0001) is issued, and synchronous establishment is not performed. Moreover, when parameter No. 8325 is set to 0, synchronous establishment is not performed.

The result of comparing the positional difference between the master axis and slave axis with a maximum allowable compensation value for synchronous establishment can be checked using the synchronous establishment enable state output signal SYNOF (F0211).

- First synchronous establishment after power-up

Two methods of performing the first synchronous establishment after power-up are available. One method is based on manual reference position return operation, and the other is based on absolute position detection.
Until this synchronous establishment is completed, synchronous error compensation is disabled. However, a synchronous error check is made.

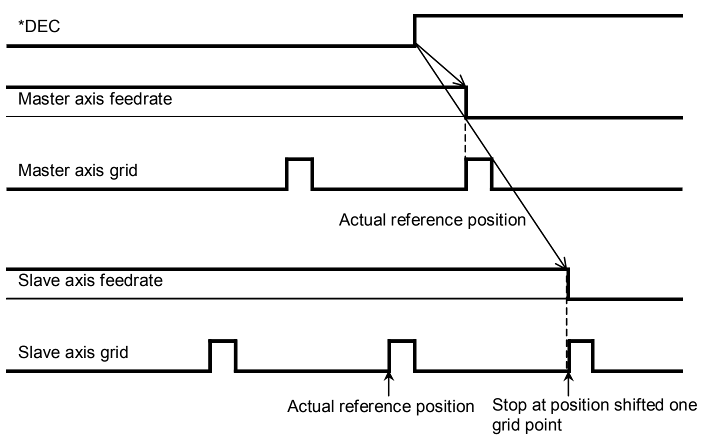
- Synchronous establishment based on manual reference position return operation

When manual reference position return operation is performed along axes under axis synchronous control, the machine is placed at the reference position on the master axis and slave axis according to the same sequence as for normal reference position return operation.
When synchronous error compensation is used, the synchronous error counter is reset and synchronous error compensation is started upon completion of reference position return along both axes.
The sequence is the same as the grid method for one axis only. However, only the deceleration signal for the master axis is used. When the deceleration signal is set to 0, the machine gradually stops along the master axis and slave axis, then an FL feedrate is set. When the deceleration signal is set to 1, the machine moves to a grid point along each of the master axis and slave axis, then stops.

NOTE

When the grid position difference between the master axis and slave axis is large, a reference position shift can occur, depending on the timing of the *DEC signal set to 1. In the example below, the shift along the slave axis is so large that the position shifted one grid point from the actual reference position is regarded as the reference position.

(Example) When the reference position on the slave axis is shifted one grid point



In such a case, match the grid position according to Subsection 19.1.4, "Automatic Setting for Grid Position Matching."

- Synchronous establishment based on absolute position detection

When an absolute-position detector is used as the position detector, the machine positions on the master axis and slave axis are found at power-up time for automatic establish synchronization.

- Synchronous establishment after emergency stop cancellation, etc.

Synchronous establishment is also performed when servo position control is turned on, for example, at emergency stop cancellation, servo alarm cancellation, or servo-off cancellation time.
However, synchronous establishment is not performed at the time of axis removal cancellation. So, synchronous establishment based on manual reference position return operation is required as in the case of power-up time.

- One-direction synchronous establishment

When synchronous error compensation is disabled, synchronous establishment can be performed by setting bit 0 (SSO) of parameter No. 8305 to 1 to move the machine in one direction along the master axis and slave axis. The move direction depends on the reference position setting based on bit 0 (SSA) of parameter No. 8304. When SSA = 0, for example, the machine coordinate on the master axis or slave axis, whichever larger, is used as the reference point. So, the machine moves in the + direction along the axes.
When bit 1 (SSE) of parameter No. 8305 is set to 1, normal synchronous establishment is performed instead of one-direction synchronous establishment after an emergency stop.

20.1.4 Automatic Setting for Grid Position Matching

Explanation

Before axis synchronous control can be performed, the reference position on the master axis must be matched with the reference position on the slave axis. With this function, the CNC automatically matches the reference positions (grid positions) on the master axis and slave axis under axis synchronous control.

[Operation procedure]

The procedure below is usable when bit 0 (ATE) of parameter No. 8303 is set to 1.

1. Set bit 1 (ATS) of parameter No. 8303 to 1.
2. Turn off the power then turn on the power.
3. Set the REF mode (or JOG mode in the case of reference position setting without dogs) when synchronous operation is ready, and make movements in the reference position return direction along the master axis and slave axis.
4. The movements along the master axis and slave axis automatically stop, and a grid difference value is set in parameter No. 8326. At this time, bit 1 (ATS) of parameter No. 8303 is set to 0, and the power-off request alarm (PW0000) is issued.
5. Turn off the power then turn on the power again.
6. Perform normal reference position return operation.

NOTE

1 Parameter setting
When bit 1 (ATS) of parameter No. 8303 is set, bit 4 (APZ) of parameter No. 1815 and parameter No. 8326 for the master axis and slave axis are set to 0. When the operator sets parameter No. 8326 (MDI, G10L50), bit 0 (ATE) of parameter No. 8303 is set to 0.

2 This function cannot be used together with the reference position shift function.

20.1.5 Synchronous Error Check

Explanation

A synchronous error value is monitored at all times. If an error exceeding a certain limit is detected, an alarm is issued and the movement along the axis is stopped.
When synchronous error compensation is performed, a check considering a positional deviation as well is made.
When synchronous error compensation is not performed, a synchronous error check based on machine coordinates is made.

- Check made when synchronous error compensation is performed

When synchronous error compensation is performed, a check considering a positional deviation is made. The actual machine position shift considering a servo positional deviation as well is checked. Depending on the value of a synchronous error, one of two alarms is issued: an alarm (DS alarm) for deceleration stop, and an alarm (SV alarm) for turning off the servo system immediately. This check is enabled when a value other than 0 is set in the parameters (Nos. 8331, 8332) for specifying a maximum allowable synchronous error. When using this method of checking a synchronous error value, disable "synchronous error check based on machine coordinates" described later by setting parameter No. 8314 to 0.

- Synchronous error excessive alarm 1 (DS0002)

When a synchronous error exceeding the value set in parameter No. 8331 is detected, synchronous error excessive alarm 1 is issued. When synchronous error excessive alarm 1 is issued, the motor gradually stops. At this time, synchronous error compensation remains enabled, so that the synchronous error is reduced by compensation. Accordingly, as the synchronous error value becomes smaller than the maximum allowable value, the alarm can be reset. If the alarm cannot be reset, the synchronous error needs to be manually corrected by selecting the correction mode described later.

- Synchronous error excessive alarm 2 (SV0002)

When a synchronous error exceeding the value set in parameter No. 8332 is detected, synchronous error excessive alarm 2 is issued. Before synchronous establishment is performed at power-up time, a value obtained by multiplying the value set in parameter No. 8332 by the coefficient set in parameter No. 8330 is used for judgement. When synchronous error excessive alarm 2 is issued, the motor immediately stops as in the case of other servo alarms. Accordingly, the positional difference between the master axis and slave axis remains uncorrected, so that the alarm cannot be reset usually. In this case, the synchronous error needs to be manually corrected by selecting the correction mode described later.

- Synchronous error check based on machine coordinates

When synchronous error compensation is not performed, a synchronous error check based on machine coordinates is made.
The machine coordinate on the master axis is compared with that on the slave axis. When the error between the machine coordinates exceeds the value set in parameter No. 8314, the SV0005 alarm is issued, and the motor is stopped immediately.
This synchronization error check can also be made in the emergency stop, servo-off, and servo alarm off state.
If a synchronous error check is made when the mode of operation is switched between synchronous operation and normal operation with an input signal, an error check is made even in normal operation. So, even if the axis synchronous control selection signal (SYNCx) or the axis synchronous control manual feed selection signal (SYNCJx) is set to 0 by mistake during synchronous operation, damage to the machine can be prevented.
The machine coordinates on the master axis and slave axis can be checked using the machine coordinate match state output signal SYNMT (F0210).

- Synchronous error check based on a positional deviation value

The servo positional deviation value of the master axis and slave axis is monitored during axis synchronous control. When the positional deviation value exceeds the limit value set in parameter No. 8323, the DS0001 alarm is issued, and the axis synchronous control positional deviation error alarm signal (F403.0) is output.
The DS0001 alarm is issued to the master axis and slave axis.
When bit 4 (SYA) of parameter No. 8301 is set to 1, the positional deviation limit value of the master axis and slave axis is checked even if a servo-off occurs during axis synchronous control.

20.1.6 Methods of Alarm Recovery by Synchronous Error Check

Explanation

To recover from an alarm issued as a result of synchronous error check, two methods are available. One method uses the correction mode, and the other uses normal operation.
If the mode of operation is switched between synchronous operation and normal operation by using an input signal, only the method using normal operation can be used.
If synchronous operation is used at all times, only the method using the correction mode can be used.

- Procedure for correcting a synchronous error by using the correction mode

Use this method if synchronous operation is used at all times without using an input signal (when bit 5 (SCA) of parameter No. 8304 is set to 1).
When the correction mode is used, synchronous error check can be temporarily disabled, and a movement can be made along the master axis or slave axis to correct a synchronous error.
In the correction mode, synchronous error compensation and error check are not performed, so that an alarm (DS0003) is issued as a warning.

1. Select the correction mode, and select an axis along which a movement is to be made by manual master axis feed. Set bit 2 (ADJ) of parameter No. 8304 of the master axis or slave axis to 1 to set the correction mode. Thus, by manual master axis feed, a movement can be made along the axis with this parameter set to 1.
   When this parameter is set to 1, the DS0003 (axis synchronous control correction mode) alarm is issued.

2. Reset the synchronous error excessive alarm.
   In this state, synchronous error compensation and error check are not performed. Be careful.

3. Select the manual mode (jog, incremental feed, or handle).

4. While checking the synchronous error value, make a movement along the master axis or slave axis in the direction that reduces the error.
   If one master axis has multiple slave axes, an attempt to reduce the synchronous error of one slave axis by master axis movement may increase the synchronous error of another slave axis, thus disabling a movement in any direction. In such a case, by setting bit 4 (MVB) of parameter No. 8304 to 1, a movement can be made in a direction that increases the synchronous error.

5. When the synchronous error is reduced to within the allowable value for suppressing the alarm, reset the value of bit 2 (ADJ) of parameter No. 8304 to the original value to switch from the correction mode to the normal synchronization mode.

Synchronous error compensation and synchronous error check are restarted.

6. Reset the correction mode alarm.

- Method of recovery using normal operation

Use this method when switching between synchronous operation and normal operation by using an input signal.
Use the procedure below for recovery from alarm SV0005.

1. Set SYNCx/SYNCJx (with x representing a slave axis number) to 0 to select normal operation.

2. Set a value greater than the current value in the parameter No. 8314 for specifying a maximum allowable synchronous error, then reset the alarm.

3. Make a movement along the master axis or slave axis by using the manual handle so that the machine coordinates of the master axis and slave axis match to a maximum possible extent.

4. Return the value of parameter No. 8314 for specifying a maximum allowable synchronous error to the original value.

20.1.7 Axis Synchronous Control Torque Difference Alarm

Explanation

If a movement made along the master axis differs from a movement made along the slave axis during axis synchronous control, the machine can be damaged. To prevent such damage, the torque command difference between the two axes is observed. If the difference is abnormal, a servo alarm (SV0420) can be issued.

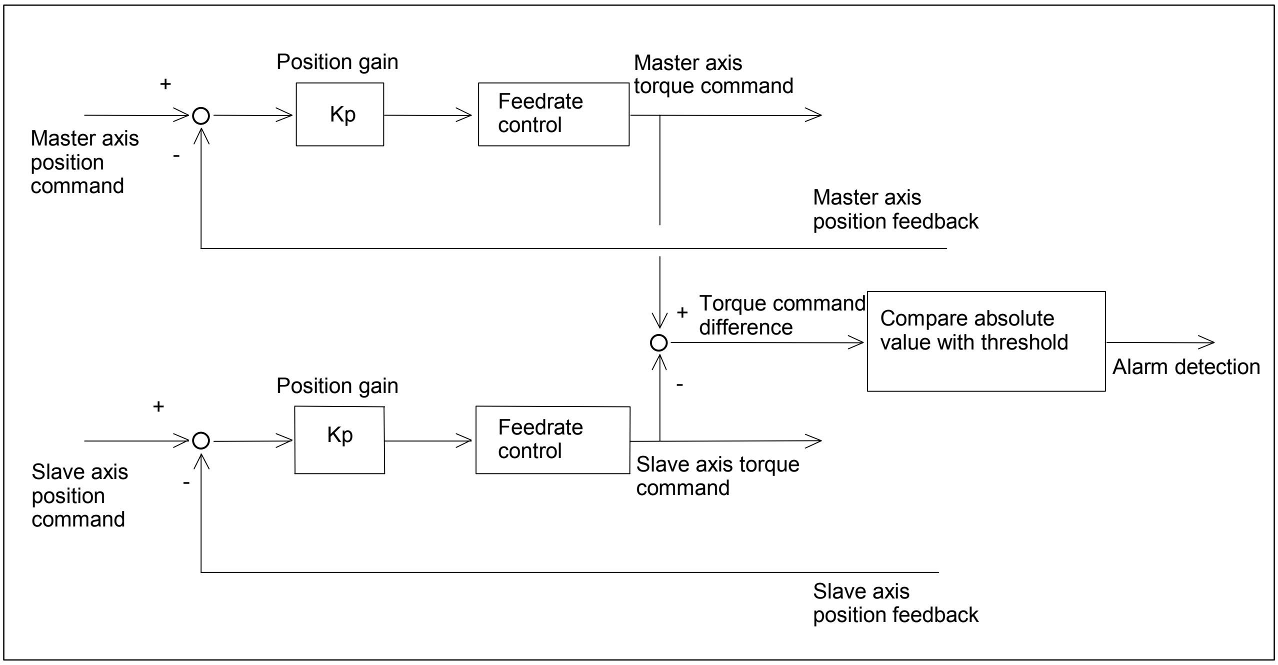


Fig. 20.1.7 (a) System configuration

[Method of use]
Specify the threshold parameter (No. 2031) according to the procedure below.

1. Set 0 in parameter No. 2031, and disable the torque difference alarm detection function.

2. To check the absolute value of the torque difference between the synchronous axes, set the parameters below. Set the same value for the two axes placed under axis synchronous control.
   Parameter No. 2115 = 0
   Parameter No. 2151 = 178

3. Display the diagnostic screen by pressing the function key <SYSTEM> then the [DGNOS] soft key.
   Diagnose No. 3500 indicates the absolute value of the torque difference between the two axes.

4. Read the absolute torque difference value presented when normal operation is being performed. In the threshold parameter (No. 2031), set a value obtained by adding some margin to the read absolute value.
   The absolute torque difference value can be observed with the Servo Guide Mate.

- Enabling/disabling of alarm detection

Alarm detection is enabled when the time set in parameter No. 8327 has elapsed after the servo ready signal SA (F000#6) is set to 1. When the input signal NSYNCA (G059.7) is set to 1, alarm detection is disabled.

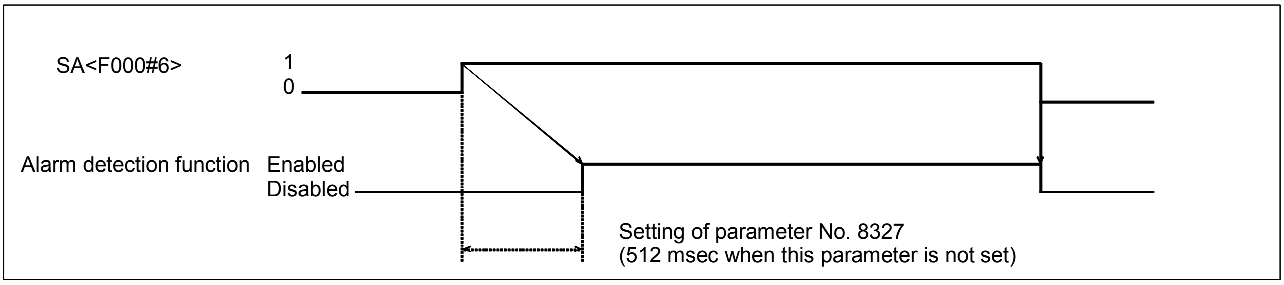


Fig. 20.1.7 (b) Timing chart

When the servo ready signal SA (F000#6) is set to 0, torque difference alarm detection is disabled.

NOTE

The servo axis number combination of the master axis and slave axis synchronized with each other must be such that an odd servo axis number is assigned to the master axis and the next servo axis number is assigned to the slave axis like (1,2) and (3,4).

⚠ CAUTION

1 When making a synchronous error check, ensure that the reference position on the master axis and the reference position on the slave axis must be at the same position.
2 In manual reference position return operation, the same operation is performed along the master axis and slave axis until a deceleration operation starts. After a deceleration operation starts, grid detection is performed for the master axis and slave axis independently of each other.
3 Pitch error compensation and backlash compensation are performed for the master axis and slave axis independently of each other.

NOTE

1 During axis synchronous control, a movement based on the reference position return check (G27), automatic reference position return (G28), 2nd/3rd/4th reference position return (G30), or machine coordinate system selection (G53) command is made as described below according to the setting of bit 7 (SRF) of parameter No. 8304.
   <1> When SRF = 0, the same movement as made along the master axis is made along the slave axis.
   <2> When SRF = 1, a movement is made along the slave axis to the specified position independently of a movement made along the master axis to the specified position.

2 A command not involving a movement along an axis such as the workpiece coordinate system setting command (G92) and local coordinate system setting command (G52) is set with the master axis according to the master axis programming.

3 During synchronous operation, the axis-by-axis signals such as for external deceleration, interlock, and machine lock are enabled for the master axis only. During synchronous operation, those signals for the slave axis are ignored.

4 When switching the synchronization state in a program, be sure to specify M codes (parameter No. 8337 and No. 8338) for turning synchronization on and off. By switching between the input signals SYNCx (G138) and SYNCJx (G140) from the PMC with the M codes, the synchronization state can be switched in the program.

5 When controlled axis removal is performed, the synchronization state is cancelled. When performing controlled axis removal, perform removal for the master axis and slave axis at the same time.

6 If a programmed command is specified for the slave axis during synchronous operation, an alarm (PS0213) is issued.
   A programmed command can be specified for the slave axis when switching between synchronous operation and normal operation is set to 0 (with bit 5 (SCA) of parameter No. 8304 set to 0) to select normal operation.

7 Axis synchronous control and PMC axis control cannot be used at the same time.

8 Axis synchronous control cannot be used with an axis for which three-dimensional coordinate conversion is enabled.

20.2 POLYGON TURNING (G50.2, G51.2)

Polygon turning means machining a workpiece to a polygonal figure by rotating the workpiece and tool at a certain ratio.

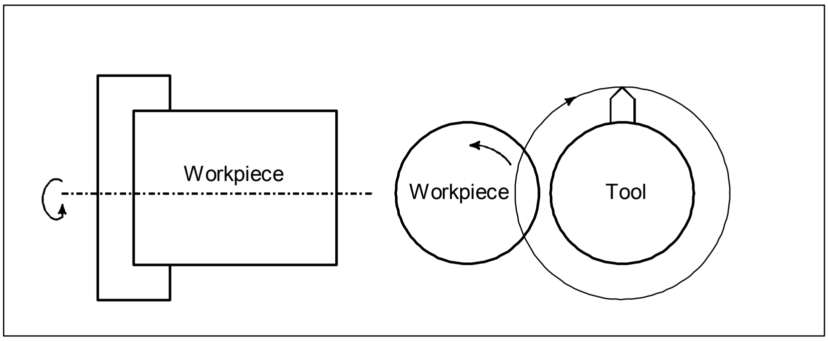


Fig. 20.2 (a) Polygon turning

By changing conditions which are rotation ratio of workpiece and tool and number of cutters, the workpiece can be machined to a square or hexagon. The machining time can be reduced as compared with polygonal figure machining using the polar coordinate interpolation. The machined figure, however, is not exactly polygonal. Generally, polygon turning is used for the heads of square and/or hexagon bolts or hexagon nuts.
As the tool rotary axis, one of the following can be used:

- CNC controlled axis (servo axis)
- Second spindle (Two serial spindles are connected.)

Polygonal machining performed using a servo axis as the tool rotary axis is referred to as polygon turning. Polygonal machining performed using the second spindle as the tool rotary axis is referred to as polygon turning with two spindles.

Function name	Workpiece axis	Tool rotary axis
Polygon turning	Spindle (Either an analog spindle or serial spindle is usable. However, a detector equivalent to a position coder is required.)	Servo axis
Polygon turning with two spindles	Spindle (Serial spindle)	Spindle (Serial spindle)

Explanation

A CNC controlled axis (servo axis) is assigned to the tool rotary axis. This rotary axis of tool is called Y-axis in the following description. As the workpiece axis (spindle), either a serial spindle or analog spindle can be used.
The Y-axis is controlled by the G51.2 command, so that the ratio of the rotation speeds of the spindle (previously specified by S-command) and the tool becomes the specified ratio.
When simultaneous start is specified by G51.2, the one-rotation signal sent from the position codes set on the spindle is detected. After one-rotation signal detection, the Y-axis is controlled using the rotation ratio of the spindle and Y-axis specified by P and Q. So, a position coder needs to be attached to the spindle. This control will be maintained until the polygon turning cancel command is executed (G50.2). Polygon turning is cancelled by any of the following in addition to the G50.2 command:

(1) Power off
(2) Emergency stop
(3) Servo alarm
(4) Reset (external reset signal ERS, reset/rewind signal RRW, and RESET key on the MDI panel)
(5) Occurrence of alarms PS0217 to PS0221, PS0314, and PS05018

NOTE

1 Before polygon turning, reference position return operation on the Y-axis needs to be specified to determine the rotation start position of the tool. This reference position return operation is performed by detecting a deceleration limit as in the case of manual reference position return operation. (By setting bit 7 (PLZ) of parameter No. 7600, reference position return operation can be performed without detecting a deceleration limit.)
2 The rotation direction on the Y-axis is determined by the sign of Q, and is not affected by the rotation direction of the position coder.
3 Among Y-axis position indications, the indication of a machine coordinate (MACHINE) changes within the amount of movement for 0 to 1 revolution as a movement is made on the Y-axis. The absolute coordinate and relative coordinate are not updated. So, when specifying an absolute-position command for the Y-axis after polygon turning mode cancellation, set a workpiece coordinate system after reference position return operation.

NOTE

4 For the Y-axis engaged in polygon turning, jog feed and handle feed are disabled.
5 For the Y-axis not engaged in polygon turning, a move command can be specified as in the case of other controlled axes.
6 The Y-axis engaged in polygon turning is not counted in the number of simultaneously controlled axes.
7 One workpiece must be machined using a fixed spindle speed until the workpiece is finished.
8 Polygon turning with two spindles cannot be used at the same time.
9 G50.2 is the G code for suppressing buffering.

⚠ **CAUTION**

1 During polygon turning, threading cannot be performed.
2 For the Y-axis engaged in synchronous operation, the signals below are valid or invalid:
Signals valid for the Y-axis
- Machine lock
- Servo-off

Signals invalid for the Y-axis
- Feed hold
- Interlock
- Override
- Dry run
(At dry run time, however, the one-rotation signal is not awaited in a G51.2 block.)

Format

G50.2	Polygon turning cancel

G51.2 P_ Q_ ;

P,Q: Rotation ratio of spindle and Y-axis

Specify range:

P: Integer from 1 to 999

Q: Integer from -999 to -1 or from 1 to 999

When Q is a positive value, Y-axis makes positive rotation.

When Q is a negative value, Y-axis makes negative rotation.

NOTE

Specify G50.2 and G51.2 in a single block.

Example

G00 X100. 0 Z20.0 S1000.0 M03 ;	
	(Workpiece rotation speed 1000 (min^{-1}))
G51.2 P1 Q2 ;	(Tool rotation start (tool rotation speed 2000 (min^{-1}))
G01 X80.0 F10.0 ;	(X-axis infeed)
G04 X2.0 ;	(Waiting 2 seconds)
G00 X100.0 ;	(X-axis escape)
G50.2 ;	(Tool rotation stop)
M05 S0 ;	(Spindle stop)

- Principle of polygon turning

The principle of polygon turning is explained below. In the figure below the radius of tool and workpiece are A and B, and the angular speeds of tool and workpiece are α and β. The origin of XY Cartesian coordinates is assumed to be the center of the workpiece.

Simplifying the explanation, consider that the tool center exists at the position

Po (A, 0) on the workpiece periphery, and the tool nose starts from position Pto (A-B, 0).

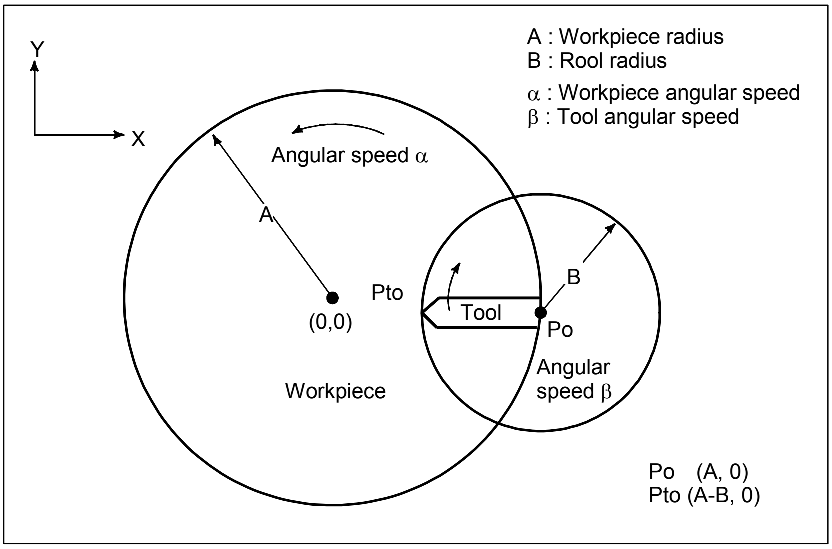


Fig. 20.2 (b) Principle of polygon turning

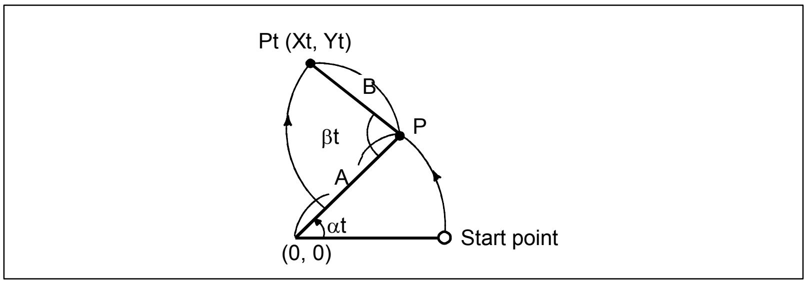


Fig. 20.2 (c) Tool nose position

In this case, the tool nose position Pt (Xt, Yt) after time t is expressed by equations 1 and 2:

$Xt = A\cos\alpha t - B\cos(\beta-\alpha)t$ (Equation 1)

$Yt = A\sin\alpha t + B\sin(\beta-\alpha)t$ (Equation 2)

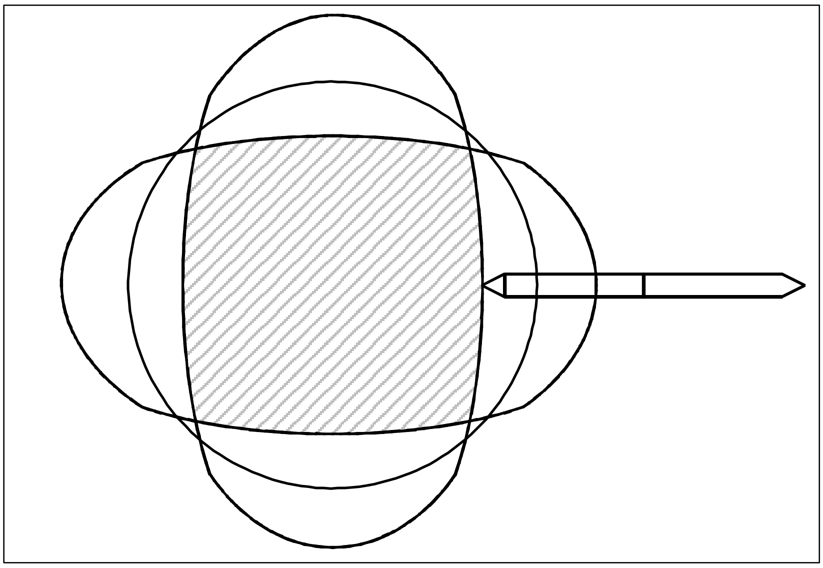
Assuming that the rotation ration of workpiece to tool is 1:2, namely, $\beta=2\alpha$, equations 1 and 2 are modified as follows:

$Xt = A\cos\alpha t - B\cos\alpha t = (A-B)\cos\alpha t$ (Equation 1)'

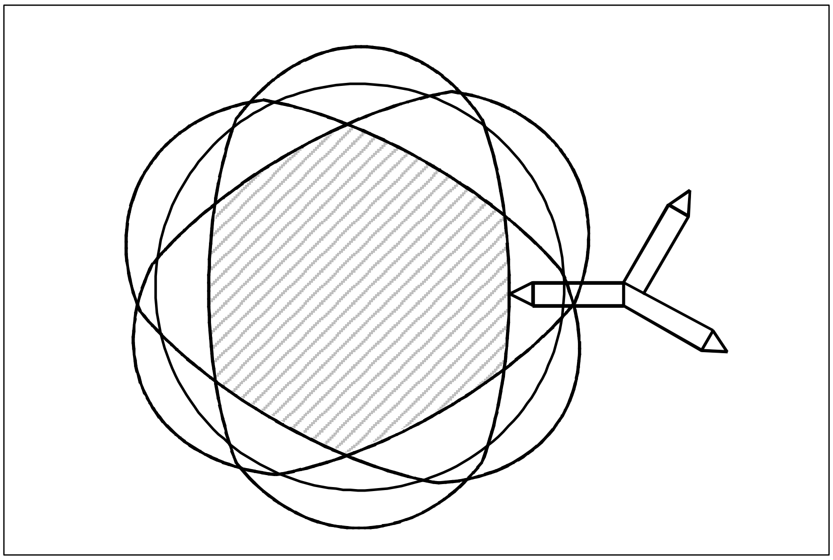
$Xt = A\sin\alpha t + B\sin\alpha t = (A+B)\sin\alpha t$ (Equation 2)'

These equations indicate that the tool nose path draws an ellipse with longer diameter A+B and shorter diameter A-B.

Then consider the case when one tool is set at 180° symmetrical positions, for a total of two. A square can be machined with these tools as shown below.



If three tools are set at every 120°, the machining figure will be a hexagon as shown below.



⚠ **WARNING**

For the maximum rotation speed of the tool, see the instruction manual supplied with the machine. Do not specify a spindle speed higher than the maximum tool speed or a ratio to the spindle speed that results in a speed higher than the maximum tool speed.

20.3 ROTARY AXIS ROLL-OVER

The roll-over function prevents coordinates for the rotary axis from overflowing. The roll-over function is enabled by setting parameter ROAx (No. 1008#0) to 1.

20.3.1 Rotary Axis Roll-over

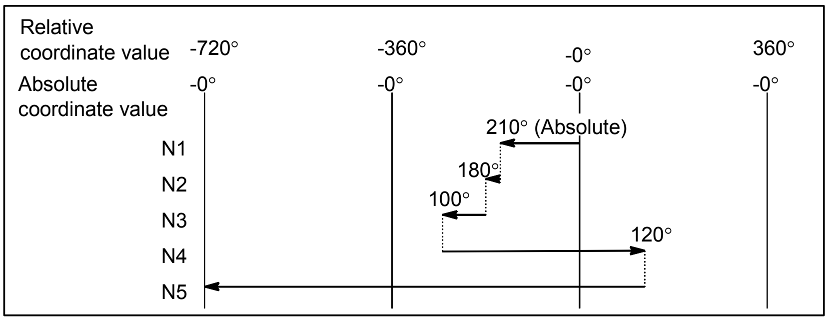
Explanation

For an incremental programming, the tool moves the angle specified in the command. For an absolute programming, the coordinates after the tool has moved are values set in parameter No. 1260, and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coordinates are closest when parameter RABx (No. 1008#1 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when parameter RRLx (No. 1008#2) is set to 1.

Example

Assume that axis A is the rotary axis and that the amount of movement per rotation is 360.000 (parameter No. 1260 = 360000). When the following program is executed using the roll-over function of the rotary axis, the axis moves as shown below.

G90 A0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 G90 A-150.0 ;	N1	-150	210
N2 G90 A540.0 ;	N2	-30	180
N3 G90 A-620.0 ;	N3	-80	100
N4 G91 A380.0 ;	N4	+380	120
N5 G91 A-840.0 ;	N5	-840	0



M

> **NOTE**
> This function cannot be used together with the index table indexing function.

20.3.2 Rotary Axis Control

This function controls a rotary axis as specified by an absolute command.
With this function, the sign of the value specified in the command is interpreted as the direction of rotation, and the absolute value of the specified value is interpreted as the coordinates of the target end position.

Explanation

This function is valid when rotary axis roll-over is enabled (parameter ROAx (No. 1008#0) is set to 1).
If the parameter RAAx (No.1007#3) is set to 1, an absolute command specified for the roll-over rotary axis is interpreted as follows: The sign and absolute value of the value specified in the command represent the direction of rotation and the end position of movement respectively.
If the parameter RAAx (No.1007#3) is set to 0, the setting by the RABx (No. 1008#1) becomes significant.

NOTE

1 This function can be used only when the corresponding option is provided. (When a machining center system is used, this function cannot be used together with the index table indexing function.)
2 This function is valid for a roll-over rotary axis.
3 If the RAAx bit (bit 3 of parameter 1007) is set to 1, the RABx bit (bit 1 of parameter 1008) is ignored. To select a rotary motion of a shorter travel distance, set both RAAx and RABx to 0.
4 This function is not supported when the machine coordinate system of the PMC axis control function is selected.

20.4 ANGULAR AXIS CONTROL

Overview

When the angular axis installed makes an angle other than 90° with the perpendicular axis, the angular axis control function controls the distance traveled along each axis according to the inclination angle as in the case where the angular axis makes 90° with the perpendicular axis.
Arbitrary axes can be specified as a set of an angular axis and perpendicular axis by parameter setting.
The actual distance traveled is controlled according to an inclination angle. However, a program, when created, assumes that the angular axis and perpendicular axis intersect at right angles. The coordinate system used at this time is referred to as the program coordinate system. (The program coordinate system may be referred to as the Cartesian coordinate system, and the actual move coordinate system may be referred to as the angular coordinate system or machine coordinate system.)

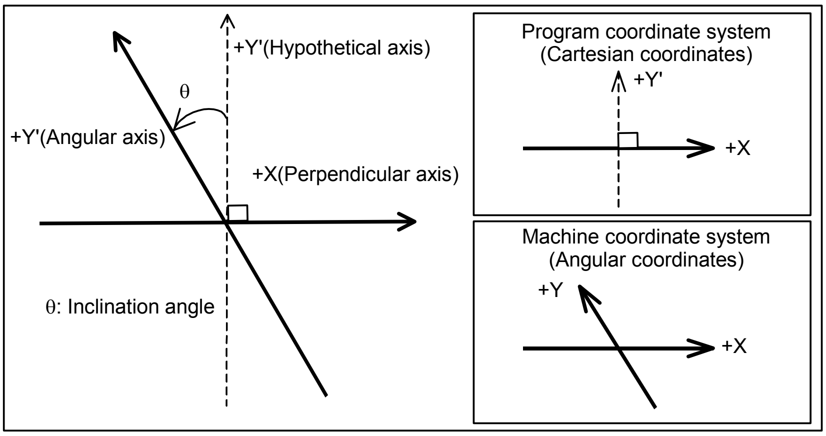


Fig. 20.4 (a)

Explanation

When the amounts of travel along the angular axis and the perpendicular axis are Ya and Xa, respectively, the amounts are controlled according to the formulas shown below.

$$Ya = \frac{Yp}{\cos\theta}$$

Xa,Ya: Actual distance
Xp,Yp: Programmed distance

The amount of travel along the perpendicular axis is corrected by the influence of travel along the angular axis, and is determined by the following formula:

$$Xa = Xp - C \times Yp \times \tan\theta$$

> **NOTE**
> The coefficient C is 1/2 in the case of diameter specification for the perpendicular axis (X) or 1 in the case of radius specification.

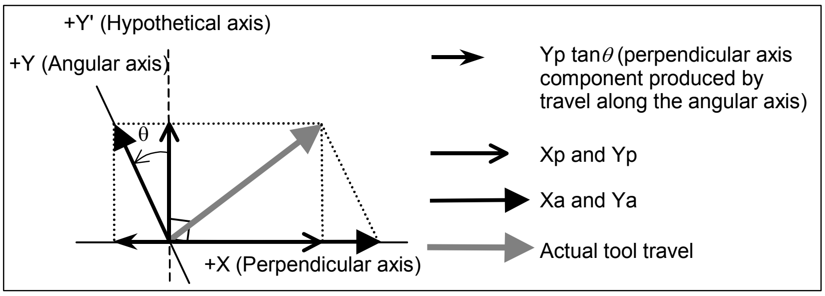


Fig. 20.4 (b)

- Feedrate

When the Y-axis is an angular axis, and the X-axis is a perpendicular axis, the feedrate along each axis is controlled as described below so that the feedrate in the tangent direction becomes Fp.
The feedrate component along the Y-axis is determined by the following expressions:

$$Fay = \frac{Fp}{\cos\theta}$$

Fa represents the actual feedrate.
Fp represents a programmed feedrate.

$$Fax = Fp - Fp \times \tan\theta$$

- Absolute and relative position display

An absolute and a relative position are indicated in the programmed Cartesian coordinate system.

- Machine position display

A machine position indication is provided in the machine coordinate system where an actual movement is taking place according to an inclination angle.

Method of use

The angular and perpendicular axes for which angular axis control is to be applied must be specified beforehand, using parameters (Nos. 8211 and 8212). When 0 is set in one of the parameters, the same number is specified in the parameters, or a number other than the controlled axis numbers is specified in a parameter, however, an angular axis and perpendicular axis are selected according to the table below.

	Angular axis	Perpendicular axis
M series	Y-axis of the basic three axes (axis with 2 set in parameter No. 1022)	Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)
T series	X-axis of the basic three axes (axis with 1 set in parameter No. 1022)	Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)

- Parameter AAC (No. 8200#0) enables or disables the angular axis control. If the function is enabled, the distance traveled along each axis is controlled according to an angular angle parameter (No. 8210).

- By using bit 2 (AZR) of parameter No. 8200, whether to make a movement along the perpendicular axis by a movement made along the angular axis when a manual reference position return operation is performed along the angular axis can be chosen. When a movement along the perpendicular axis is enabled (AZR = 1), a reference position return operation along the perpendicular axis can be performed by a movement made along the angular axis.
- By setting the normal axis/angular axis control invalid signal NOZAGC to 1, angular axis control only for the angular axis can be available. In this time the angular axis are converted to those along the angular coordinate system without affecting commands to normal axis.
  Use this signal when operating each axis independently.

- Manual reference position return operation

A movement is made to the reference position (machine position) set in parameter No. 1240. By using bit 2 (AZR) of parameter No. 8200, whether to make a movement along the perpendicular axis when a reference position return operation is performed along the angular axis can be chosen.

- Automatic reference position return operation and floating reference position return operation (G28, G30, G30.1)

A movement to the middle point along the angular axis affects a movement along the perpendicular axis. A movement from the middle point to the reference position along the angular axis does not affect a movement along the perpendicular axis, regardless of whether the perpendicular axis/angular axis control disable signal (NOZAGC) is turned on or off. If manual reference position return operation is not performed even once after the power is turned on, operation is performed in the same sequence as for manual reference position return operation. So, specify commands first for the angular axis then for the perpendicular axis.

Example) When the Y-axis is an angular axis and the X-axis is a perpendicular axis

(1) If the angular axis is first specified then the perpendicular axis is specified, reference position return operation is performed normally.
G28Y_;
G28X_;

(2) If the perpendicular axis is first specified then the angular axis is specified, or if the perpendicular axis and the angular axis are specified at the same time, alarm PS0372 is issued when a movement is made along the perpendicular axis.

$\left\{\begin{array}{l}\text{G28X\_;}\\\text{G28Y\_;}\end{array}\right.$ or $\left\{\text{G28X\_ Y\_;}\right.$

- Reference position return operation of high-speed type

When a reference position is already established and a reference position return operation of high-speed type is to be performed, the reference position return operation need not be performed in the order from the angular axis to the perpendicular axis.

- Machine coordinate selection (G53)

By specifying (G90)G53X_Y_: (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°), a movement is made by rapid traverse.
However, a movement along the angular axis (G53 command) does not affect a movement along the perpendicular axis, regardless of whether the perpendicular axis/angular axis control disable signal (NOZAGC) is turned on or off.
Example)

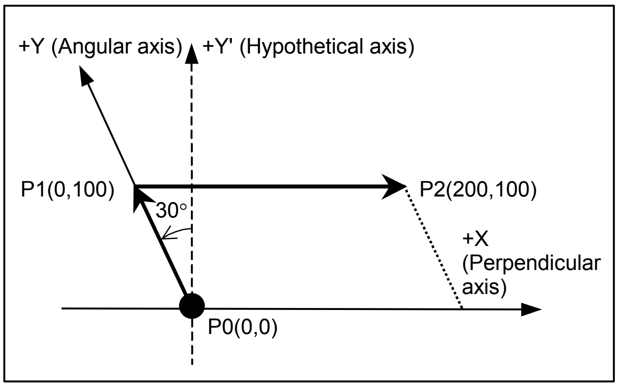
1 Move command for movement from point P0 to point P1
  >G90G53Y100.
2 Move command for movement from point P1 to point P2
  >G90G53X200.

<1> Coordinates of P1

(Absolute coordinates)		(Machine coordinates)	
X	-50.000	X	0.000
Y	86.603	Y	100.000

<2> Coordinates of P2

(Absolute coordinates)		(Machine coordinates)	
X	150.000	X	200.000
Y	86.603	Y	100.000



- Commands for linear interpolation and linear interpolation type positioning (G01, G00)

The tool moves to a specified position in the Cartesian coordinate system when the following is specified:
(G90)G00X_Y_; (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°)
or
(G90)G01X_Y_F_; (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°)

Example) Examples of positioning

1 Move command for movement from point P0 to point P1
>G90G00Y100.

2 Move command for movement from P1 to P2
G90G00X200.

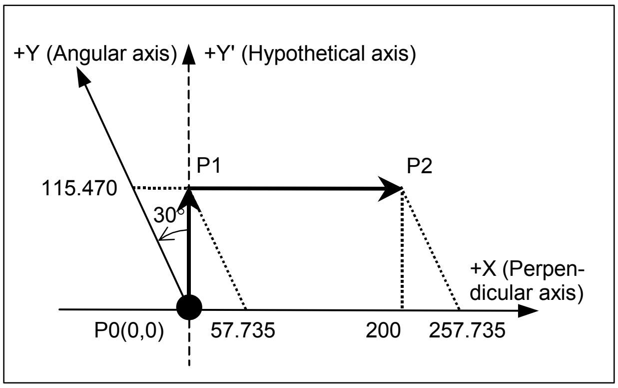
(1) When the perpendicular axis/angular axis control disable signal (NOZAGC) is set to 0

<1> Coordinates of P1

(Absolute coordinates)		(Machine coordinates)	
X	0.000	X	57.735
Y	100.000	Y	115.470

<2> Coordinates of P2

(Absolute coordinates)		(Machine coordinates)	
X	200.000	X	257.735
Y	100.000	Y	115.470



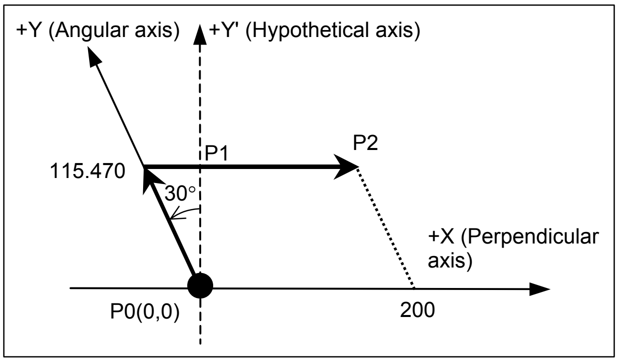
(2) When the perpendicular axis/angular axis control disable signal (NOZAGC) is set to 1

<1> Coordinates of P1

(Absolute coordinates)		(Machine coordinates)	
X	0.000	X	0.000
Y	100.000	Y	115.470

<2> Coordinates of P2

(Absolute coordinates)		(Machine coordinates)	
X	200.000	X	200.000
Y	100.000	Y	115.470



- Three-dimensional coordinate conversion

In the three-dimensional coordinate conversion mode, angular coordinate system conversion is applied to the workpiece coordinate system that has undergone three-dimensional coordinate conversion.

- Stored stroke limit

Stored stroke limits under angular axis control can be set not in a angular coordinate system but in the Cartesian coordinate system by setting bits 0, 1, and 2 (AOT, AO2, and AO3) of parameter No. 8201.

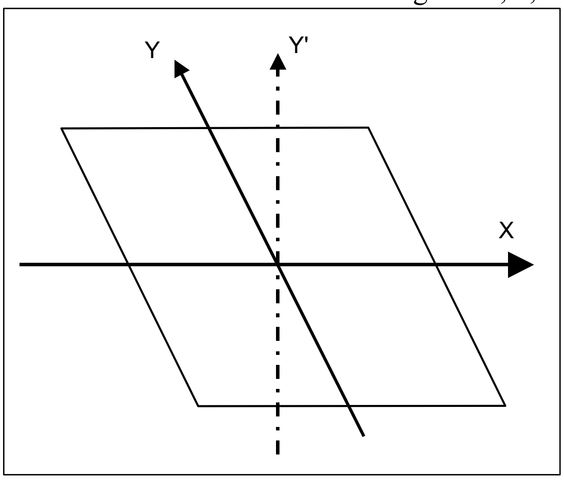


Fig. 20.4 (c) OT area in a angular coordinate system

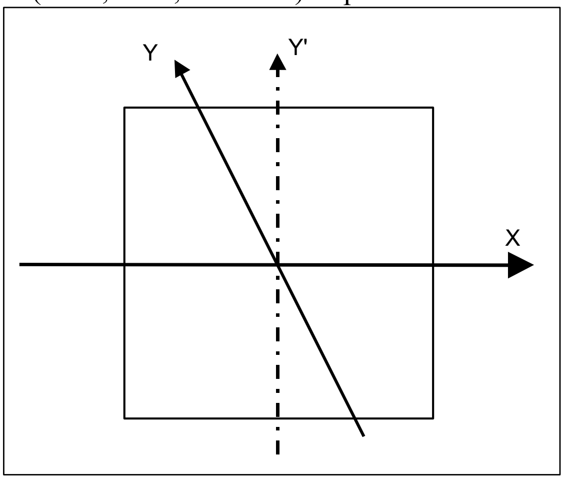


Fig. 20.4 (d) OT area in a Cartesian coordinate system

Machine coordinates include a value converted for the angular axis and a compensation value for the perpendicular axis, so that a angular machine coordinate system as shown in Fig. 20.4 (c) results.
A stored stroke limit is checked in the machine coordinate system, so that the limit area is slanted to form a rhombus as shown in Fig. 20.4 (c). In this case, the area cannot be identified intuitively. So, stroke limits are checked not in an actual angular machine coordinate system but in a virtual Cartesian machine coordinate system as shown in Fig. 20.4 (d).

The functions that operate in the Cartesian coordinate system are:

- Stored stroke check 1 (Both of I and II)
- Stored stroke check 2 (G22/G23)
- Stored stroke check 3
- Stored stroke check before move

The stored stroke check function before move does not work in a angular coordinate system. Unless this function is enabled, and the coordinate system is converted to the Cartesian coordinate system, no stroke check is made.

- Stroke limit external setting (function specific to the M series only and valid only for OT1)
- Bit 7 (BFA) of parameter No. 1300 for specifying whether to issue an alarm before or after a stroke limit is exceeded (valid for OT1 and OT3)

The stored stroke limit functions other than the above work in a angular coordinate system.

- Relationships between this function and axis-by-axis input/output signals

The table below indicates the relationships between this function and the meaning of each controlled axis signal.
The input/output signals are classified as signals valid for the program coordinate system (Cartesian coordinate system) and signals valid for the machine coordinate system (angular coordinate system). In the "Classification" column, "Cartesian" is indicated for a signal that is valid for the Cartesian coordinate system, and "Angular" is indicated for a signal that is valid for the angular coordinate system.
A signal valid for the Cartesian coordinate system means a signal valid for a specified axis, and a signal valid for the angular coordinate system is a signal valid for actual machine movement.
This means that when a movement is made along the perpendicular axis by a movement along the angular axis alone:

A signal valid for the Cartesian coordinate system is affected by a movement along the angular axis.
A signal valid for the angular coordinate system is not affected by a movement along the angular axis.

Input signal				
Signal name		**Address**	**Classification**	**Remarks**
Interlock for each axis	*ITx	G130	Cartesian	When a movement is made along the angular axis only, interlocking the perpendicular axis does not interlock a movement along the perpendicular axis made by a movement along the angular axis. Caution) When using the interlock signal for each axis, make both of the angular axis and perpendicular axis high.
Overtravel	*+Lx *-Lx	G114 G116	Angular	This signal is applied to each axis independently. (If the perpendicular axis is made high, no alarm is issued for the perpendicular axis even when an OT alarm is issued for the angular axis.)
Deceleration signal for reference position return	*DECx	X009	Angular	This signal is applied to each axis independently.
Servo-off signal	SVFx	G126	Angular	This signal is applied to each axis independently.
Control axis detach signal	DTCHx	G124	Angular	This signal is applied to each axis independently.
Feed axis direction selection signal	+Jx -Jx	G100 G102	Cartesian	A movement is made in the Cartesian coordinate system. (When the +J/-J signal for the angular axis is made high, a movement is made also along the perpendicular axis.)

Input signal				
Signal name		**Address**	**Classification**	**Remarks**
Mirror image	MIx	G106	Angular	Mirror image is applied to the angular coordinate system for each axis independently. Caution) Be sure to turn off the mirror image signal for the angular axis and perpendicular axis engaged in manual operation.
Manual feed interlock signal for each axis direction, tool compensation value write signal	+MIT1, +MIT2	X004#2, #4	Cartesian	Set the tool compensation parameter in the Cartesian coordinate system.
Machine lock for each axis	MLKx	G108	Angular	This signal is applied to each axis independently.

Output signal				
Signal name		**Address**	**Classification**	**Remarks**
In-position signal	INPx	F104	Angular	Applied to each axis independently.
Mirror image check signal	MMIx	F108	Angular	Applied to each axis independently.
Controlled axis removal in-progress signal	MDTCHx	F110	Angular	Applied to each axis independently.
Travel in-progress signal	MVx	F102	Angular	Applied to each axis independently.
Reference position return completion signal	ZPx	F094	Cartesian	Applied to each axis independently. (A manual reference position return operation and the first automatic reference position return operation after power-up need to be performed first for the angular axis.)
2nd reference position return completion signal	ZP2x	F096	Cartesian	Applied to each axis independently.
3rd reference position return completion signal	ZP3x	F098	Cartesian	Applied to each axis independently.
4th reference position return completion signal	ZP4x	F100	Cartesian	Applied to each axis independently.

Limitation

- Three-dimensional coordinate conversion

If the basic three axes in the three-dimensional coordinate conversion mode do no include a perpendicular axis and angular axis for angular axis control, operation cannot be performed normally in a correct angular coordinate system.

- Linear scale with absolute address reference mark

- For both of the angular axis and perpendicular axis, a linear scale with an absolute address reference mark must be used.
- Reference position return operation must be first completed along the angular axis.
- Return operation cannot be performed along the perpendicular axis while return operation is being performed along the angular axis.

- Synchronous control

For synchronous control on axes related to angular axis control, the angular axis and Cartesian axis on the master axis side and the angular axis and Cartesian axis on the slave axis side must be placed under synchronous control at the same time. Moreover, synchronous control can be exercised between angular axes only or between Cartesian axes only.
If an attempt is made to perform operation under a condition other than the above, the alarm (PS0375) is issued.
Example)

Path 1		Path 2
X1 (Cartesian axis)	←Synchronous→	X2 (Cartesian axis)
Y1 (angular axis)	←Synchronous→	Y2 (angular axis)

- Mixture control

For mixture control on axes related to angular axis control, the angular axis and Cartesian axis on the master axis side and the angular axis and Cartesian axis on the slave axis side must be placed under mixture control at the same time. Moreover, mixture control can be exercised between angular axes only or between Cartesian axes only.
If an attempt is made to perform operation under a condition other than the above, the alarm (PS0375) is issued.
Example)

Path 1		Path 2
X1 (Cartesian axis)	←mixture→	X2 (Cartesian axis)
Y1 (angular axis)	←mixture→	Y2 (angular axis)

- Rigid tapping

As a rigid tapping axis, no angular axis can be used.

- Functions that cannot be used simultaneously

- Axis synchronous control, twin table control, parallel axis control, polygon turning, hypothetical axis control, EGB function, PMC axis control, superimposed control

⚠ CAUTION

1 After angular axis control parameter setting, be sure to perform manual reference position return operation.
2 Before manual reference position return operation is performed along the perpendicular axis, reference position return operation along the angular axis must be completed (with the reference position return completion signal for the angular axis (ZPx) set to 1). If reference position return operation is performed along the perpendicular axis first, an alarm PS5460 is issued.
3 When the setting is made so that the tool moves along the perpendicular axis during manual reference position return along the angular axis (bit 2 (AZK) of parameter No. 8200 is set to 0), if once manual reference position return has been performed along the angular axis, also perform manual reference position return along the perpendicular axis immediately after the operation. Alarm PS0090 is issued when an attempt is made to manually return to the reference position along the perpendicular axis although the angular axis is not on the reference point.
4 Before attempting to manually move the tool along the angular and perpendicular axes simultaneously, set perpendicular/angular axis control disable signal NOZAGC to 1.
5 Once the tool has been moved along the angular axis when perpendicular/angular axis control disable signal NOZAGC has been set to 1, manual reference position return must be performed.
6 The same increment system must be used with the angular axis and perpendicular axis.
7 Before a perpendicular axis reference position return check can be made, angular axis reference position return operation must be completed.
8 No rotary axis must be set for the angular axis and perpendicular axis. A rotary axis may be specified only for a linear axis.
9 Set a position switch operation range (parameter Nos. 6930 to 6965) in a angular coordinate system.

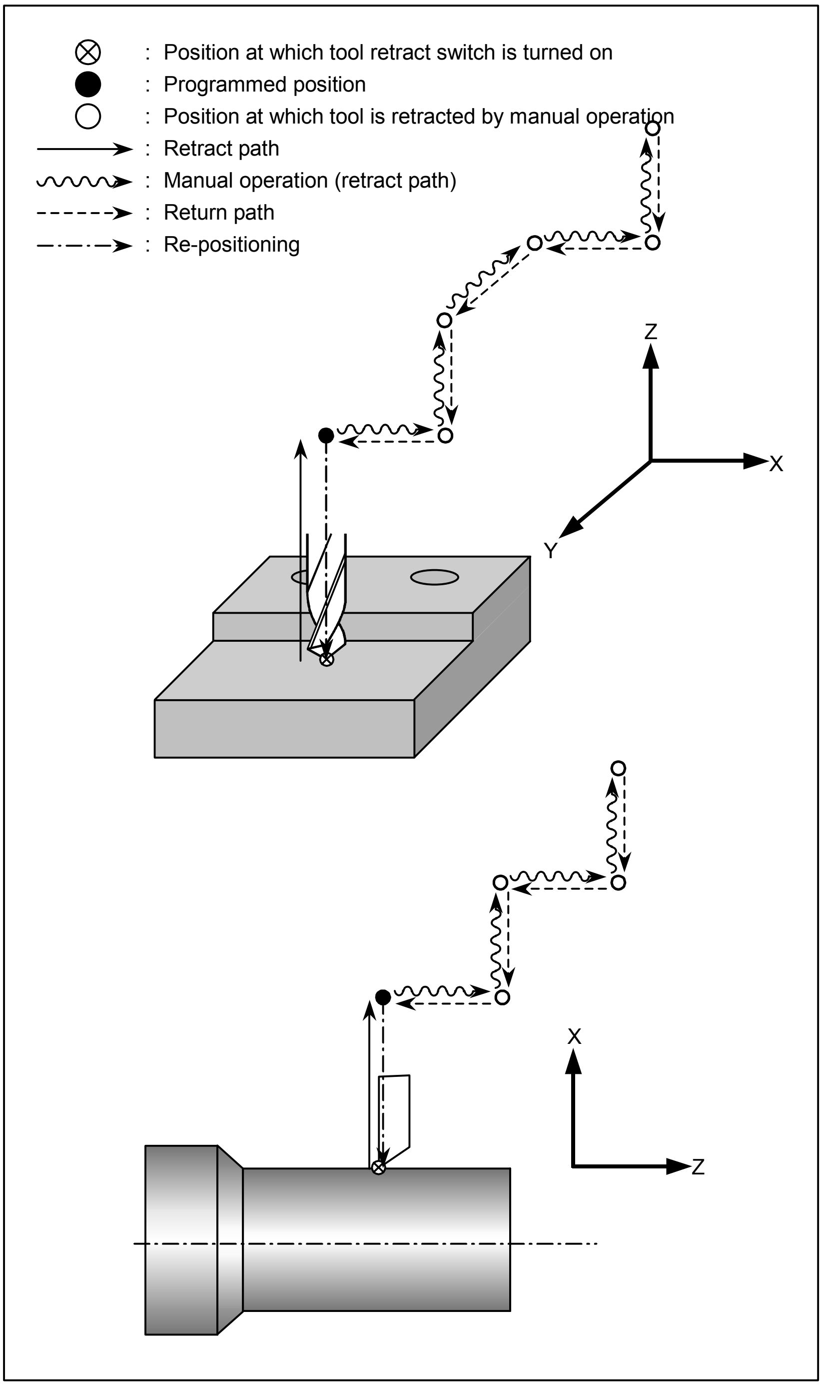
20.5 TOOL RETRACT AND RECOVER

Overview

To replace the tool damaged during machining or to check the status of machining, the tool can be withdrawn from a workpiece. The tool can then be advanced again to restart machining efficiently.
The tool retract and recover operation consists of the following four steps:

- Retract
  The tool is retracted to a predefined position using the TOOL WITHDRAW switch.
- Withdrawal
  The tool is moved to the tool-change position manually.
- Return
  The tool returns to the retract position.
- Repositioning
  The tool returns to the interrupted position.

For the tool retract and recover operations, see “Tool retract and recover ” in the Part III.


: Position at which tool retract switch is turned on
: Programmed position
: Position at which tool is retracted by manual operation
: Retract path
: Manual operation (retract path)
: Return path
: Re-positioning
Z
X
Y
X
Z

Format

Specify a retraction axis and distance in the following format:

Specify the amount of retraction, using G10.6.

G10.6 IP_ ;

IP: In incremental mode, retraction distance from the position where the retract signal is turned on
In the absolute mode, retraction distance to an absolute position
The specified amount of retraction is effective until G10.6 is next executed. To cancel the retraction, specify the following:
G10.6 ; (as a single block containing no other commands)

Explanation

- Retraction

When the TOOL WITHDRAW switch on the machine operator's panel is turned on during automatic operation or in the automatic operation stop or hold state, the tool is retracted the length of the programmed retraction distance. This operation is called retraction. The position at which retraction is completed is called the retraction position. The tool is retracted to the retraction position in linear interpolation at the dry run feedrate. When retraction is completed, the RETRACTIONS POSITION LED on the machine operator's panel goes on.
When the TOOL WITHDRAW switch is turned on during execution of a block in automatic operation, execution of the block is interrupted immediately and the tool is
retracted. After retraction is completed, the system enters the automatic operation hold state.
If the retraction distance and direction are not programmed, retraction is not performed. In this state, the tool can be withdrawn and returned.
When the TOOL WITHDRAW switch is turned on in the automatic operation stop or hold state, the tool is retracted, then the automatic operation stop or hold state is entered again.
When the TOOL WITHDRAW switch is turned on, the tool withdraw mode is set. When the tool withdraw mode is set, the TOOL BEING WITHDRAWN LED on the machine operator's panel goes on.

- Withdrawal

When the manual mode is set, the tool can be moved manually (jog feed, incremental feed, handle feed, or manual numeral command) to replace the tool or measure a machined workpiece. This operation is called a withdrawal. The tool withdrawal path is automatically memorized by the CNC.

- Return

When the mode is returned to automatic operation mode and the TOOL RETURN switch on the machine operator's panel is turned off, the CNC automatically moves the tool to the retraction position by tracing the manually-moved tool path backwards. This operation is called a return. Upon completion of a return to the retraction position, the RETRACTIONS POSITION LED comes on.

- Repositioning

When the cycle start button is pressed while the tool is in the retraction position, the tool moves to the position where the TOOL WITHDRAW switch was turned on. This operation is called repositioning. Upon completion of repositioning, the TOOL BEING WITHDRAWN LED is turned off, indicating that the tool withdrawal mode has terminated. Operation after completion of repositioning depends on the automatic operation state when the tool withdrawal mode is set.

(1) When the tool withdrawal mode is set during automatic operation, operation is resumed after completion of repositioning.

(2) When the tool withdrawal mode is set when automatic operation is held or stopped, the original automatic operation hold or stop state is set after completion of repositioning. When the cycle start button is pressed again, automatic operation is resumed.

Limitation

- Offset

If the origin, presetting, or workpiece origin offset value (or External workpiece origin offset value) is changed after retraction is specified with G10.6 in absolute mode, the change is not reflected in the retraction position. After such changes are made, the retraction position must be respecified with G10.6.

When the tool is damaged, automatic operation can be interrupted with a tool withdrawal and return operation in order to replace the tool. Note that if the tool offset value is changed after tool replacement, the change is ignored when automatic operation is resumed from the start point or other point in the interrupted block.

- Machine lock, mirror image, and scaling

When withdrawing the tool manually in the tool withdrawal mode, never use the machine lock, mirror-image, or scaling function.

- Reset

Upon reset, the retraction data specified in G10.6 is cleared. Retraction data needs to be specified again.

- Retraction command

The tool withdrawal and return function is enabled even when the retraction command is not specified. In this case, retraction and repositioning are not performed.

> ⚠ **WARNING**
> The retraction axis and retraction distance specified in G10.6 need to be changed in an appropriate block according to the figure being machined. Be very careful when specifying the retraction distance; an incorrect retraction distance may damage the workpiece, machine, or tool.

20.6 ELECTRIC GEAR BOX

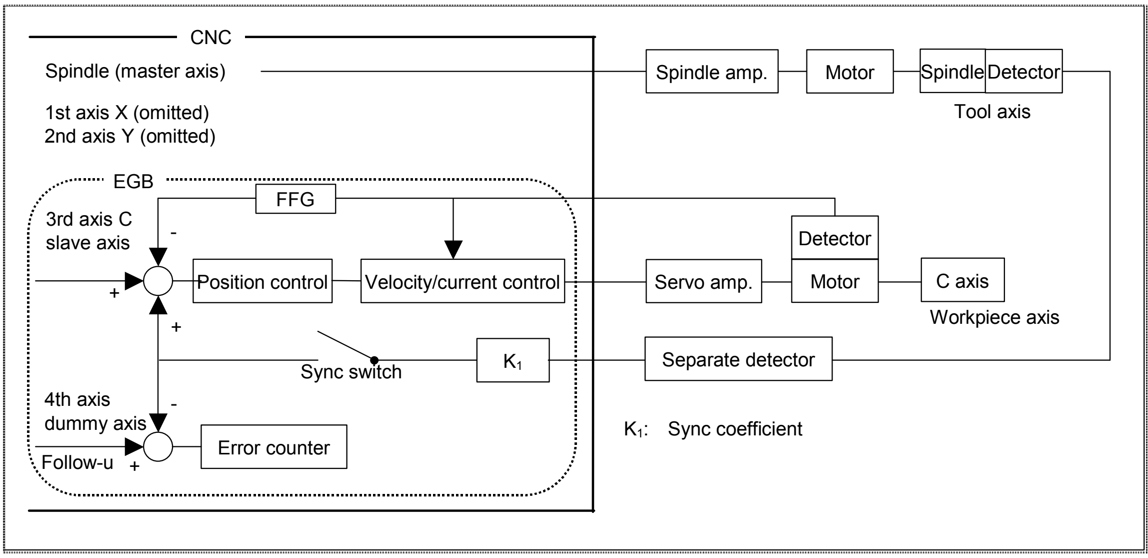
20.6.1 Electric Gear Box

Overview

This function enables fabrication of high-precision gears, screws, and other components by rotating the workpiece in synchronization with a rotating tool or by moving the tool in synchronization with a rotating workpiece. The rate of synchronization can be specified with a program. The synchronization of tool and workpiece axes with this function adopts a system in which the synchronization is directly controlled by digital servo, so that the workpiece axis can follow up the speed fluctuations on the tool axis with no error, thereby allowing fabrication of high-precision cogwheels. In the subsequent explanation, the Electronic Gear Box is called the EGB.
Some conditions must be satisfied for setting the workpiece axis and tool axis. For details, refer to the relevant manual provided by the machine tool builder.

- Example of controlled axis configuration

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis.)



> **NOTE**
> The sampling cycle in which feedback pulses are read from the master axis, the synchronization pulses of the slave axis is calculated based on the synchronization coefficient K, and the pulses are issued for the position control for the slave axis is 1 ms.

Format

G81 T_ (L_) (Q_ P_) ; **Starts synchronization.**
G80 ; **Ends synchronization.**

T : Number of teeth (Specifiable range: 1 to 1000)
L : Number of hob threads (Specifiable range: -21 to +21)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
If L is not specified, the number of hob threads is assumed 1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.1 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.1 to 254.0 inch^{-1})
P : Gear helix angle
(Unit: 0.00001deg, Specifiable range: -90.0 to 90.0deg)
* When specifying Q and P, the user can use a decimal point.

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.
Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.

A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

- Synchronization control

(1) Start of synchronization
If G81 is issued so that the machine enters synchronization mode, the synch switch of the EGB function is closed, and the synchronization of the tool and workpiece axes is started. During synchronization, the rotation about the tool and workpiece axes is controlled so that the relationship between T (number of teeth) and L (number of hob threads) is maintained. During synchronization, the synchronization relationship is maintained regardless of whether the operation is automatic or manual.
Specify P and Q to use helical gear compensation.
If only either P or Q is issued, alarm PS1594 is generated.
If, during synchronization, G81 is issued again without synchronization cancelation, alarm PS1595 is generated if ECN, bit 3 of parameter No. 7731, is 0. If ECN, bit 3 of parameter No. 7731, is 1, helical gear compensation is conducted with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued, and if T and L commands are not issued and only P and Q commands are issued, helical gear compensation is conducted with the synchronization coefficient kept intact. This allows consecutive fabrication of helical gears and super gears.

(2) Start of tool axis rotation
When the rotation of the tool axis starts, the rotation of the workpiece starts so that the synchronous relationship specified in the G81 block can be maintained.
The rotation direction of the workpiece axis depends on the rotation direction of the tool axis. That is, when the rotation direction of the tool axis is positive, the rotation direction of the workpiece axis is also positive; when the rotation direction of the tool axis is negative, the rotation direction of the workpiece axis is also negative. However, by specifying a negative value for L, the rotation direction of the workpiece axis can be made opposite to the rotation direction of the tool axis.
During synchronization, the machine coordinates of the workpiece axis and EGB axis are updated as synchronous motion proceeds. On the other hand, a synchronous move command has no effect on the absolute and relative coordinates.

(3) Termination of tool axis rotation
In synchronism with gradual stop of the tool axis, the workpiece axis is decelerated and stopped. By specifying the command below after the spindle stops, synchronization is canceled, and the EGB synchronization switch is opened.

(4) Cancellation of synchronization

When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization. Subsequently, absolute commands for the workpiece axis will be enabled.

For a rotation axis, the amount of travel during synchronization, as rounded to 360-degree units is added to the absolute coordinate.

In the G80 block, only O and N addresses can be specified.

By setting HBR, bit 0 of parameter No. 7700, to 0, it is possible to cancel synchronization with a reset. If the manual absolute signal is ON, the absolute coordinate is updated.

Synchronization is automatically canceled under the following conditions:

<1> An emergency stop is applied.
<2> A servo alarm is generated.
<3> An overtravel (software or hardware) alarm is generated for the slave axis.
<4> Alarm PW0000 (POWER MUST BE OFF) is generated.
<5> An IO alarm is generated.

CAUTION

1 During synchronization, a move command can be issued for the workpiece and other axes, using a program. The move command for the workpiece and other axes must, however, be an incremental one.
2 In synchronization mode, G27, G28, G29, G30, G30.1, G33, or G53 cannot be specified for the workpiece axis.
3 Controlled axis detach cannot be used for the workpiece axis.

NOTE

1 During synchronization, manual handle interruption can be performed on the workpiece and other axes.
2 The maximum feedrates on the tool and workpiece axes are restricted by the position detector used.
3 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
4 In synchronization mode, only the machine coordinate on the slave axis is updated.
5 If EFX, bit 0 of parameter No. 7731, is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set EFX, bit 0 of parameter No. 7731, to 1 and use G81.8 instead of G81 and G80.8 instead of G80.

> **NOTE**
> 6 If TDP, bit 0 of parameter No. 7702, is 1, the permissible range of T is 0.1 to 100 (1/10 of the specified value).
> 7 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if LZR, bit 3 of parameter No.7701, is 0; if LZR, bit 3 of parameter No.7701, is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
> 8 Feed per revolution is performed on the feedback pulses on the spindle. By setting ERV, bit 0 of parameter No. 7703, to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
> 9 In EGB synchronization mode, AI contour control mode is temporarily canceled.

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

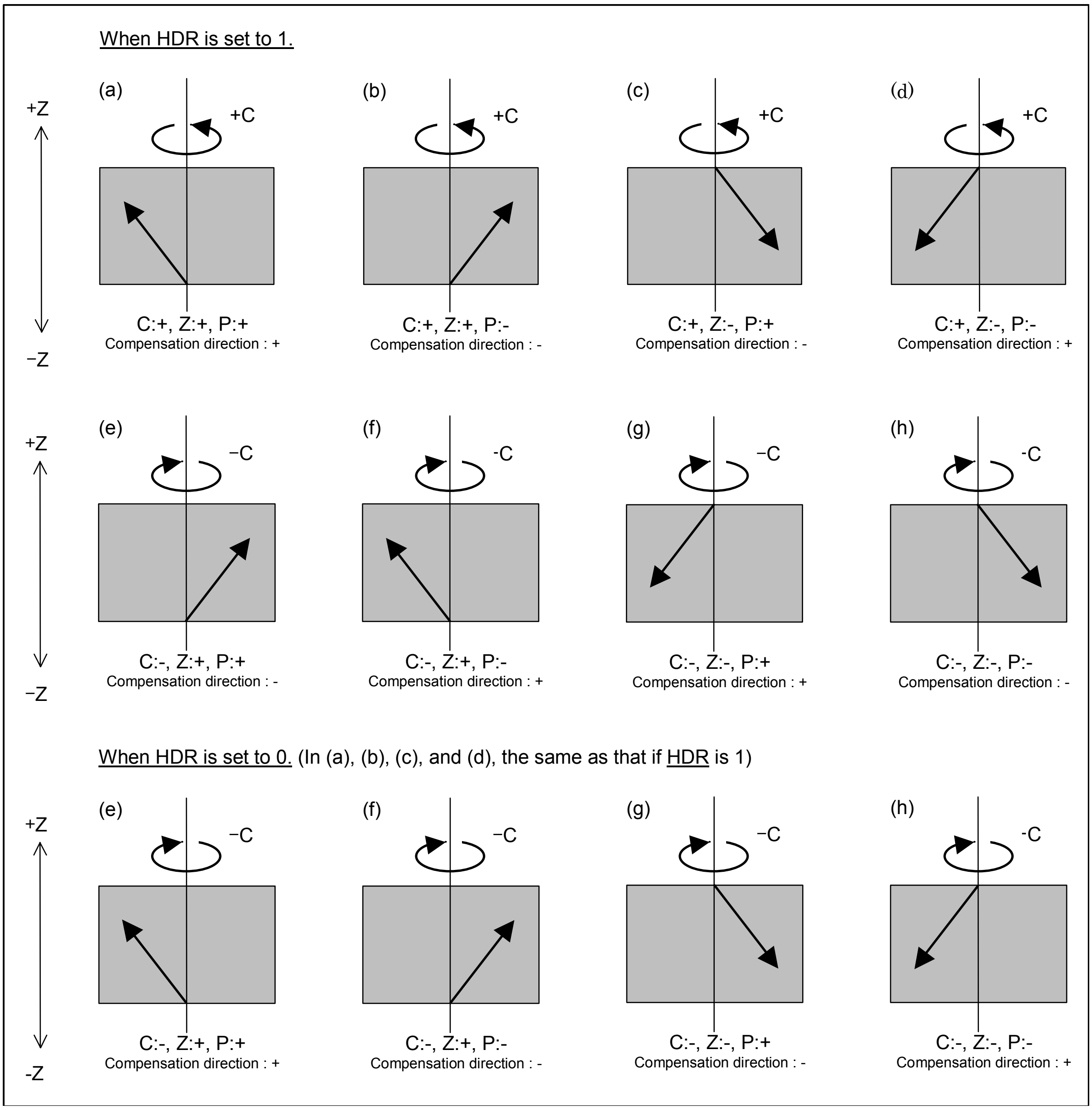
where

Compensation angle: Signed absolute value (deg)
Z : Amount of travel on the Z-axis after the specification of G81
P : Signed gear helix angle (deg)
π : Ratio of the circumference of a circle to its diameter
T : Number of teeth
Q : Module (mm) or diametral pitch (inch-1)
Use P, T, and Q specified in the G81 block.

In helical compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical compensation.

- Direction of helical gear compensation

The direction depends on HDR, bit 2 of parameter No. 7700.



- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (Kn/Kd) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_n}{K_d} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where

L : Number of teeth

T : Number of hob threads

α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772)

β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773)

Kn / Kd is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:

$-2147483648 \leq K_n \leq 2147483647$

$1 \leq K_d \leq 65535$

When this restriction is not satisfied, the alarm (PS1596)is issued when G81 is specified.

Example

O1000 ;	
N0010 M19 ;	Tool axis orientation
N0020 G28 G91 C0 ;	Reference position return on the workpiece axis
N0030 G81 T20 L1 ;	Synchronous start on tool and workpiece axes (Rotation about the workpiece axis by 18° per rotation about the tool axis)
N0040 S300 M03 ;	Rotation about the tool axis at 300min^{-1}
N0050 G01 X__F__;	Movement along the X-axis (cut)
N0060 G01 Z__F__;	Movement along the Z-axis (machining)
-----------------------------;	If necessary, axis commands such as C, X, and Z commands are allowed.
N0100 G01 X__F__;	Movement along the X-axis (escape)
N0110 M05 ;	Stop on the tool axis
N0120 G80 ;	Synchronous cancellation on tool and workpiece axes
N0130 M30 ;	

- Retract function

(1) Retract function with an external signal

When the retract switch on the machine operator's panel is turned on, retraction is performed with the retract amount set in parameter No. 7741 and the feedrate set in parameter No. 7740.
No movement is performed along an axis for which 0 is set as the retract amount.
For the retract switch, refer to the relevant manual provided by the machine tool builder.

(2) Retract function with a servo spindle alarm

If, during EGB synchronization or automatic operation, a CNC alarm is generated due to an error on the servo or spindle axis, retraction is performed with the retract amount set in parameter No. 7741 and the speed set in parameter No. 7740.
This can prevent the tool and the object being machined from damage if a servo alarm is generated.

- Conditions under the retract function with a servo spindle alarm can be executed

  The conditions under which the retract function with a servo spindle alarm can be executed can be changed using the settings of ARE, bit 1 of parameter No. 7703, and ARO, bit 2 of parameter No. 7703.
  The table below lists parameter settings and corresponding conditions.

ARE	ARO	Condition
1	0	EGB synchronization is in progress.
1	1	Both EGB synchronization and automatic operation are in progress.
0	0	Either EGB synchronization or automatic operation is in progress.
0	1	

⚠ CAUTION

1. Retraction is performed at the speed specified in parameter No. 7740.
2. Feed hold is not effective to movement during retraction.
3. Feedrate override is not effective to movement during retraction.

NOTE

1. During a retract operation, an interlock is effective to the retract axis.
2. During a retract operation, a machine lock is effective to the retract axis.
3. When the retract switch is turned on during automatic operation, retraction is performed and automatic operation is held.
4. During retraction, automatic operation cannot be performed.
5. Retract movement is performed with non-linear type positioning.
6. To enable the retract function with a servo spindle alarm, ART, bit 3 of parameter No.7702, must be set.
7. After the end of retraction with a servo alarm, servo position control stops in 400 ms.

20.6.2 Electronic Gear Box Automatic Phase Synchronization

Overview

In the electric gear box (EGB), when synchronization start or cancellation is specified, synchronization is not started or canceled immediately. Instead, acceleration/deceleration is executed first. Therefore, synchronization can be started or canceled while the spindle is rotating. Also, synchronization ratio can be changed while the spindle is rotating.

At synchronization start, automatic phase synchronization is performed such that the workpiece axis position matches the position corresponding to the spindle one-rotation signal. With this synchronization, the same operation is performed as synchronization start caused by a one-rotation signal in hobbing synchronization when using the functions of a hobbing machine.

The spindle corresponds to the EGB master axis and the workpiece axis corresponds to an EGB slave axis.

Format

- Acceleration/deceleration type

G81 T _ L _ R1 ; Synchronization start
G80 R1 ; Synchronization cancellation

T : Number of teeth (range of valid settings: 1-1000)
L : Number of hob threads (range of valid settings: -21 to +21, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

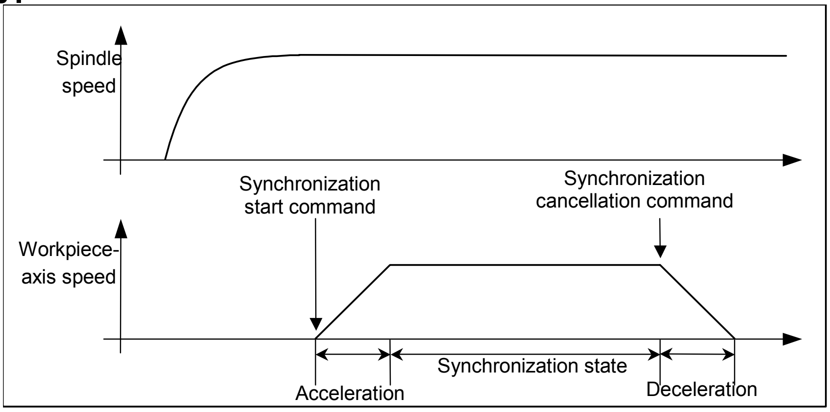
- Acceleration/deceleration plus automatic phase synchronization type

G81 T _ L _ R2 ; Synchronization start
G80 R1 ; Synchronization cancellation

T : Number of teeth (range of valid settings: 1-1000)
L : Number of hob threads (range of valid settings: -21 to +21, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

Explanation

- Acceleration/deceleration type



1. Specify G81R1 to start synchronization.
   When G81R1 is specified, the workpiece axis (slave axis) is subject to acceleration at the acceleration rate set in parameter No. 7778. When the synchronization speed is reached, the G81R1 block is terminated.
2. For cancellation, specify G80R1 while the tool is moved away from the workpiece.
3. When G80R1 is specified, deceleration is started immediately at the acceleration rate set in parameter No. 7778.
   When the speed is reduced to 0, the G80R1 block is terminated.

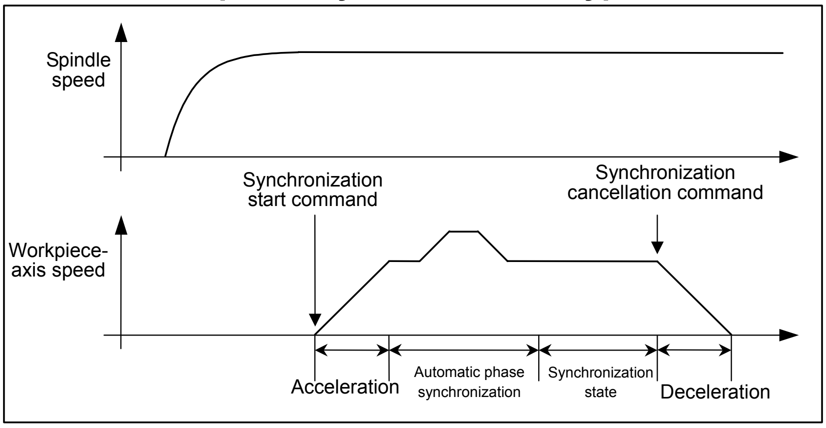
NOTE

1 During synchronization start/cancellation, acceleration/deceleration is linear.

2 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization is canceled:
<1> Reset
<2> PW0000 (POWER MUST BE OFF)
<3> IO alarm

3 If EFX, bit 0 of parameter No. 7731, is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set EFX, bit 0 of parameter No. 7731, to 1 and use G81.8 instead of G81 and G80.8 instead of G80.

- Acceleration/deceleration plus automatic phase synchronization type



1. Move the workpiece axis to the position that corresponds to that of the one-rotation signal of the spindle.
2. Specify G81R2 to start synchronization.
   When G81R2 is specified, the workpiece axis is accelerated with the acceleration according to the acceleration rate set in the parameter (No.7778). Upon completion of phase synchronization, the G81R2 block terminates.
3. For cancellation, specify G80R2 while the tool is moved away from the workpiece.
4. When G80R2 is issued, deceleration is started immediately according to the acceleration rate set in parameter No. 7778. When the speed reaches 0, the G80R2 block terminates.

CAUTION

1 Set the automatic phase synchronization speed in parameter No. 7776 and the movement direction in PHD, bit 7 of parameter No. 7702.

2 Phase synchronization acceleration/deceleration is performed with the rapid traverse linear acceleration/deceleration rate (time constant specified in parameter No. 1620).

3 The workpiece axis speed is the speed synchronized with spindle rotation with automatic phase synchronization speed being superposed. When setting the position deviation limit parameter (No. 1828), take the superposition into consideration.

NOTE

1 The one-rotation signal used for automatic phase synchronization is issued not by the spindle position coder but by the separate pulse coder attached to the spindle and used to collect EGB feedback information. This means that the orientation position based on the one-rotation signal issued by the spindle position coder does not match the position used as the reference for the workpiece axis when establishing phase synchronization for automatic phase synchronization based on G81R2. Moreover, the one-rotation signal of the separate pulse coder must be turned on for each rotation of the spindle.
2 With the use of parameter No.7777, the position at which the phase of the workpiece axis is matched can be shifted from the position corresponding to the one-rotation signal in automatic phase matching.
3 In automatic phase synchronization, when a synchronization command is issued again in a synchronization state, movement about the workpiece axis is performed such that the position corresponding to the one-rotation signal of the spindle matches the position about the workpiece axis specified in the G81R2 synchronization start command executed first.
4 In automatic phase synchronization, movement is performed about the workpiece axis from the current position to the nearest phase position in the phase synchronization movement direction specified by the parameter.
5 Linear acceleration/deceleration applies to synchronization start/cancellation.
6 The acceleration/deceleration plus automatic phase synchronization type can be executed by the PHS parameter, bit 6 of No.7702, without specifying an R2 command in a G81 or G80 block.
7 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization canceled:
<1> Reset
<2> PW0000 (POWER MUST BE OFF)
<3> IO alarm
8 The acceleration rate parameter (No.7778) must not be changed in the synchronization mode.
9 If EFX, bit 0 of parameter No. 7731, is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set EFX, bit 0 of parameter No. 7731, to 1 and use G81.8 instead of G81 and G80.8 instead of G80.

Program example

- Acceleration/deceleration type

M03 :;	Clockwise spindle rotation command
G81 T_ L_ R1 ;	Synchronization start command
G00 X_ ;	Positions the workpiece at the machining position.

Machining in the synchronous state

G00 X_ ;	Retract the workpiece from the tool.
G81 T_ L_ R1 ;	Synchronization ratio change.
G00 X_ ;	Positions the workpiece at the machining position.

Machining in the synchronous state

G00 X_ ;	Retract the workpiece from the tool.
G80 R1 ;	Synchronization cancel command

- Acceleration/deceleration plus automatic phase synchronization type

M03 ;	Clockwise spindle rotation command
G00 G90 C_ ;	C-axis positioning
G81 T_ L_ R2 ;	Synchronization start command
G00 X_ ;	Positions the workpiece at the machining position.

Machining in the synchronous state

G00 X_ ;	Retract the workpiece from the tool.
G81 T_ L_ R2 ;	Synchronization ratio change.
G00 X_ ;	Positions the workpiece at the machining position.

Machining in the synchronous state

G00 X_ ;	Retract the workpiece from the tool.
G80 R2 ;	Synchronization cancel command

20.6.3 Skip Function for EGB Axis

Overview

This function enables the skip or high-speed skip signal (these signals are collectively called skip signals in the remainder of this manual) for the EBG slave axis in synchronization mode with the EGB (electric gear box).
This function has features such as the following:

1. If a skip signal is input while an EGB axis skip command block is being executed, this block does not terminate until the specified number of skip signals have been input.
2. If a skip signal is input while an EGB axis skip command block is being executed, the tool remains in synchronous mode and moves, not stopping on the EGB slave axis.
3. The machine coordinates assumed when skip signals are input and the number of input skip signals are stored in specified custom macro variables.

Format

G81 T_ L_ ;	**EGB mode ON**
G31.8 G91 α0 P_ Q_ (R_) ;	**EGB skip command**

α: Specify an EGB slave axis. The specified value must always be "0".
P: Number of the first one of the custom macro variables used to store the machine coordinates assumed when skip signals are input.
Q: Number of skip signals that can be input during the execution of G31.8
(permissible range: 1 to 200).
R: Number of the custom macro variable used to store the number of input skip signals.
Specify it to check the number of input signals.

Explanation

G31.8 is a one-shot G code.
When G31.8 is executed, the machine coordinates assumed when skip signals are input are written in as many custom macro variables as the number specified in Q, starting with the one having the number specified in P, when the skip command block terminates.
Also, the number of input skip signals is written to the custom macro variable specified in R each time a skip signal is input.

Example

```
G81    T200    L2 ;                            EGB mode ON
X_  ;
Z_  ;
G31.8    G91    C0    P500    Q200    R1 ;    EGB skip command
```

After 200 skip signals have been input, the 200 skip positions on the C-axis that correspond to the respective skip signals are stored in custom macro variables #500 to #699.
Also, the number of input skip signals is stored in custom macro variable #1.

> **NOTE**
> 1 When specifying this function, specify only a single EGB slave axis. If no axis is specified for two or more axes are specified, alarm PS1152 is generated.
> 2 If P is not specified, alarm PS1152 is generated.
> 3 If R is not specified, the number of input skip signals is not written to a custom macro variable.
> 4 The custom macro variable numbers specified in P and R must be existing ones. If a non-existent variable number is specified, alarm PS0115 is generated.
> If a variable shortage occurs, alarm PS0115 is generated.
> 5 Whether to use conventional skip signals or high-speed skip signals with this function can be specified with HSS, bit 4 of parameter No. 6200. If deciding to use high-speed skip signals, specify which high-speed signals to enable with 9S1 to 9S8, bits 0 to 7 of parameter No. 6208).
> 6 Skip positions are calculated from feedback pulses from the machine. Thus, they are free from errors due to delay in acceleration/deceleration and the servo system

20.6.4 Electronic Gear Box 2 Pair

Overview

The Electronic Gear Box is a function for rotating a workpiece in sync with a rotating tool, or to move a tool in sync with a rotating workpiece. With this function, the high-precision machining of gears, threads, and the like can be implemented. A desired synchronization ratio can be programmed.
Up to two sets of axes can be synchronized. A gear grinding machine can be controlled, for instance, by using one axis for rotating the workpiece in sync with the tool and another axis for performing dressing in sync with the tool.
The specification method differs depending on the configuration of the machine. For details, refer to the relevant manual provided by the machine tool builder.
The electric gear box is hereinafter called an EGB function.

20.6.4.1 Specification method (G80.5, G81.5)

Format

$$\mathbf{G81.5}\quad \left\{\begin{matrix} \mathbf{T}\,t \\ \mathbf{P}\,p \end{matrix}\right\}\qquad \left\{\begin{matrix} \beta\, j \\ \beta 0\ \ \mathbf{L}\, l \end{matrix}\right\}\ ;\ \text{Synchronization start}$$

↑ Amount of travel along the master axis

↑ Amount of travel along the slave axis

G80.5 β0 ; Synchronization end

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.
Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.
A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

- Synchronization start

When the ratio of the master-axis travel to the slave-axis travel is specified, synchronization starts.

Specify the master-axis travel in either of the following ways.

1 Master-axis speed

T t: Master-axis speed (1≤ t ≤1000)

2 Master-axis pulse count

P p : Master-axis pulse count (1≤ p ≤999999999)

Specify a pulse count on the condition that four pulses correspond to one period in the A and B phases.

Specify the slave-axis travel in either of the following ways.

1 Slave-axis travel

βj : Slave-axis address

Slave-axis travel indicated in units of the minimum travel increments

(the range of valid settings for usual axis movement applies)

When j = 0, the specified command is regarded as being a command for the slave-axis speed, described below. In this case, if L is not specified, an alarm is output.

2 Slave-axis speed

β0 L±*l*

β : Slave-axis address

l : Slave axis speed(1≤ *l* ≤21)

⚠ CAUTION

1 A move command can be issued by a program to the slave axis or other axes during synchronization. In this case, the command shall be an incremental command.
2 A G27, G28, G29, G30, G30.1,G33 or G53 command cannot be issued to the slave axis in synchronization mode.
3 Controlled-axis detach cannot be used for the master axis or the slave axis.

NOTE

1 During synchronization, manual handle interruption can be performed on the slave and other axes.
2 The maximum feedrates on the master and slave axes are restricted by the position detector used.
3 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
4 In synchronization mode, only the machine coordinate on the slave axis is updated.

NOTE

5 If G81.5 is issued again during synchronization, alarm PS1595 is generated if ECN, bit 3 of parameter No. 7731, is 0. If ECN, bit 3 of parameter No. 7731, is 1, the synchronization coefficient can be changed to a newly specified one.

6 If EFX, bit 0 of parameter No. 7731, is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set EFX, bit 0 of parameter No. 7731, to 1.

7 In EGB synchronization mode, AI contour control mode is temporarily canceled.

- Synchronization end

1 Canceling synchronization for each axis by issuing a command

With a G80.5 β0 command, synchronization is canceled.

β is the address of the slave axis. Synchronization of the slave axis specified by β is canceled.

A cancellation command can be issued only for one axis in one block.

When β0 is not specified, the synchronization of all currently synchronized axes is canceled.

When a synchronization cancellation command is issued, the absolute coordinates for the slave axis are updated according to the amount of travel during synchronization. For a rotation axis, the value obtained by rounding off the amount of travel during synchronization to the nearest 360 degrees is added to the absolute coordinates.

2 Canceling synchronization by a reset

By setting HBR, bit 0 of parameter No. 7700, to 0, synchronization is canceled with a reset. If the manual absolute signal is ON, the absolute coordinates are updated.

3 Others

Synchronization is automatically canceled under the following conditions.

<1> Emergency stop

<2> Servo alarm

<3> A slave axis overtravel (software, hardware) alarm is generated

<4> Alarm PW0000 (indicating that the power should be turned off)

<5> An IO alarm is generated

20.6.4.2 Description of commands compatible with those for a hobbing machine (G80, G81)

A command compatible with that for a hobbing machine can be used as a synchronization command.
Usually, a hobbing machine performs machining by synchronizing the workpiece axis (usually, the C-axis) to the hobbing axis (spindle).
If there are two synchronization sets with the EGB, which synchronization set to start with this specification method can be specified with parameter No. 7710.

Format

G81 T_ (L_) (Q_ P_) ;   Synchronization start
G80 ;   Synchronization end

T : Number of teeth (range of valid settings: 1 to 1000)
L : Number of hob threads (range of valid settings: -21 to +21, excluding 0)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
If L is 0, the direction depends on LZR, bit 3 of parameter No. 7701.
When L is not specified, the number of hob threads is regarded as being 1.
Q : Module or diametral pitch
In metric input, specify a module
(unit : 0.00001mm, range of valid settings : 0.1 to 25.0mm)
In inch input, specify a diametral pitch
(unit : 0.00001inch^{-1}, range of valid settings : 0.1 to 254.0 inch^{-1})
P : Twisted angle of a gear
(unit : 0.00001deg, range of valid settings : -90.0 to 90.0deg)
* The Q and P specifications may include the decimal point.

Explanation

- Synchronization start

Specify P and Q to use helical gear compensation. In this case, if only one of P and Q is specified, alarm (PS1594) is generated.
When G81 is issued so that the machine enters synchronization mode, the synchronization of the workpiece axis to the spindle is started.
During synchronization, control is performed such that the ratio of the spindle speed to the workpiece-axis speed is the same as that of T (number of teeth) to L (number of hob threads).
If, during synchronization, G81 is issued again without synchronization cancelation, alarm PS1595 is generated if ECN, bit 3 of parameter No. 7731, is 0. If ECN, bit 3 of parameter No. 7731, is 1, helical gear compensation is conducted with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued, and if T and L commands are not issued and only P and Q commands are issued, helical gear compensation is conducted with the synchronization coefficient kept intact. This allows consecutive fabrication of helical gears and super gears.

⚠ CAUTION

1. A move command can be issued by a program to the workpiece axis or other axes during synchronization. In this case, the command shall be an incremental command.
2. A G27, G28, G29, G30, G30.1,G33 or G53 command cannot be issued to the workpiece axis in synchronization mode.
3. Controlled-axis detach cannot be used for the workpiece axis.

NOTE

1. During synchronization, manual handle interruption can be performed on the workpiece and other axes.
2. The maximum feedrates on the tool and workpiece axes are restricted by the position detector used.
3. In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
4. In synchronization mode, only the machine coordinate on the slave axis is updated.
5. If EFX, bit 0 of parameter No. 7731, is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set EFX, bit 0 of parameter No. 7731, to 1 and use G81.8 instead of G81 and G80.8 instead of G80.

> **NOTE**
> 6 If TDP, bit 0 of parameter No. 7702, is 1, the permissible range of T is 0.1 to 100 (1/10 of the specified value).
> 7 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if LZR, bit 3 of parameter No.7701, is 0; if LZR, bit 3 of parameter No.7701, is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
> 8 Feed per revolution is performed on the feedback pulses on the spindle. By setting ERV, bit 0 of parameter No. 7703, to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
> 9 In EGB synchronization mode, AI contour control mode is temporarily canceled.

- Synchronization end

Synchronization of all synchronized axes is canceled.
When a synchronization cancellation command is issued, the absolute coordinates for the slave axis are updated according to the amount of travel during synchronization.
For a rotation axis, the value obtained by rounding off the amount of travel during synchronization to the nearest 360 degrees is added to the absolute coordinates.
In a G80 block, do not specify addresses other than O or N.

- Helical gear compensation

For a helical gear, the workpiece axis is subjected to compensation for movement along the Z axis (axial feed axis) according to the twisted angle of the gear.
Helical gear compensation is performed with the following data.

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

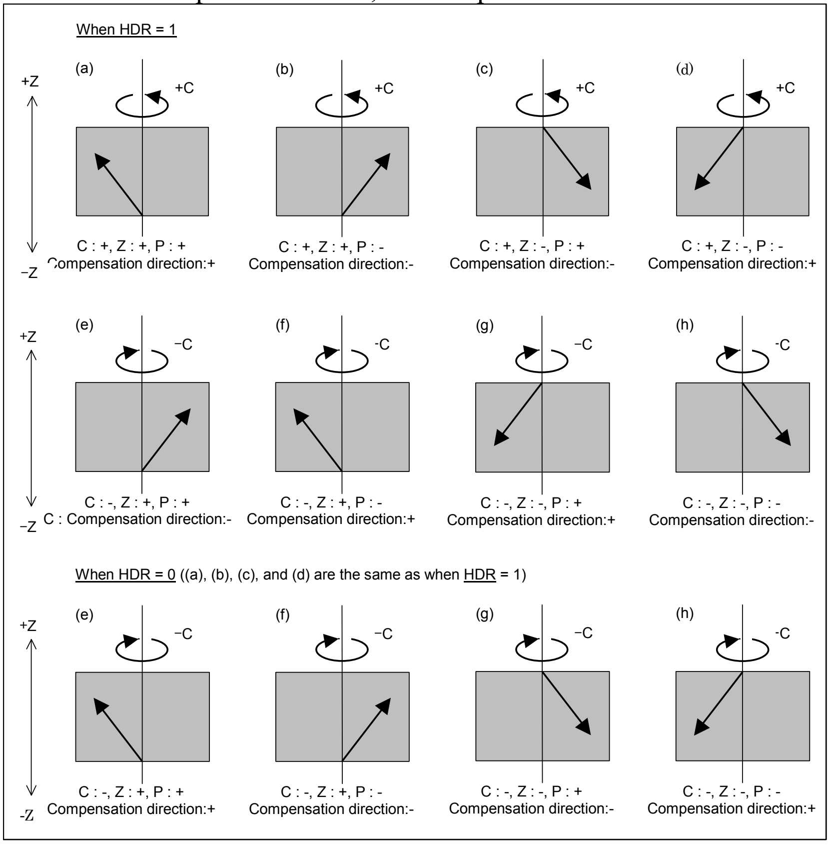
$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

where

Compensation angle: Absolute value with sign (degrees)
Z : Amount of travel along the Z axis after a G81 command is issued (mm or inch)
P : Twisted angle of the gear with sign (degrees)
π : Ratio of the circumference of a circle to its diameter
T : Number of teeth
Q : Module (mm) or diametral pitch (inch-1)
Use P, T, and Q specified in the G81 block.

- Direction of helical gear compensation

The direction depends on HDR, bit 2 of parameter No. 7700.



20.6.4.3 Controlled axis configuration example

- For gear grinders

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis)
5th axis : V axis (EGB slave axis : Dressing axis)
6th axis : V axis (EGB dummy axis : Cannot be used as a normal controlled axis)

Cannot be used as a normal controlled axis)

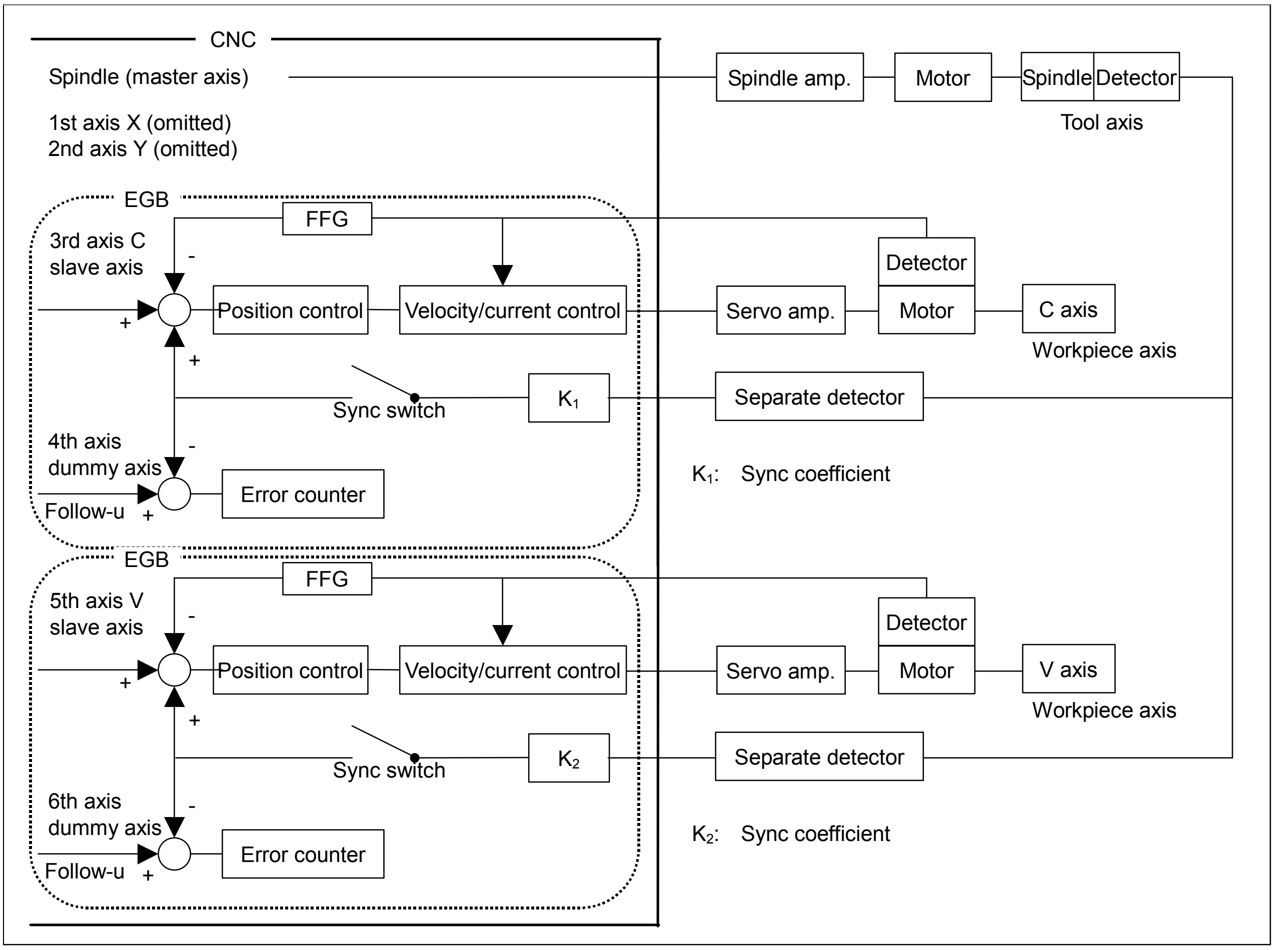


Fig. 20.6.4.3 (a)

NOTE
The sampling cycle in which feedback pulses are read from the master axis, the synchronization pulses of the slave axis is calculated based on the synchronization coefficient K, and the pulses are issued for the position control for the slave axis is 1 ms.

20.6.4.4 Sample programs

- When the master axis is the spindle, and the slave axis is the C-axis

(a) G81.5 T10 C0 L1 ;
Synchronization between the master axis and C-axis is started at the ratio of one rotation about the C-axis to ten rotations about the master axis.

(b) G81.5 T10 C0 L-1 ;
Synchronization between the master axis and C-axis is started at the ratio of one rotation about the C-axis to ten rotations about the master axis.
In this case, however, the direction of rotation is opposite to that of (a).

(c) G81.5 T1 C3.26 ;
Synchronization between the master axis and C-axis is started at the ratio of a 3.26-degree rotation about the C-axis per one rotation about the master axis.

(d) G81.5 P10000 C-0.214 ;
Synchronization between the master axis and C-axis is started at the ratio of a -0.214 degree rotation about the C-axis to 10,000 feedback pulses from the pulse coder of the master axis.

- When the master axis is the spindle, the slave axis is the V-axis (linear axis), and inch/metric conversion is performed

(a) For a millimeter machine and metric input
G81.5 T1 V1.0 ;
Synchronization between the master axis and V-axis is started at the ratio of a 1.00 mm movement along the V-axis per rotation about the master axis.

(b) For a millimeter machine and inch input
G81.5 T1 V1.0 ;
Synchronization between the master axis and V-axis is started at the ratio of a 1.0 inch movement (25.4 mm) along the V-axis per rotation about the master axis.

- When two groups of axes are synchronized simultaneously

Based on the controlled axis configuration described in Fig. 20.6.4.3 (a), the sample program below synchronizes the spindle with the V-axis while the spindle is synchronized with the C-axis.

O0100 ;		
....		
N01 Mxx ;	Performs spindle orientation.	
N02 G00 G90 C... ;	Positions the C-axis.	
N03 G81.5 T10 C0 L1 ;	Starts synchronization at the ratio of one rotation about the C-axis to ten spindle rotations.	
N04 Myy S300 ;	Rotates the spindle.	Grinding
N05 G01 G91 X... Y... F... ;	Makes movements for grinding.	
....		
N06 Mzz ;	Stops the spindle.	
....		
N10 G00 G90 V... ;	Positions the dressing axis.	Dressing
N11 G81.5 P12000 V0.5 ;	Starts synchronization at the ratio of a 0.5 mm movement along the V-axis every 12,000 pulses on the spindle.	
N12 Myy S300 ;	Rotates the spindle.	
N20 M98 P9500 ;	A subprogram for dressing is called. (Note)	
N21 Mzz ;	Stops the spindle.	
N30 G80.5 V0 ;	Cancels V-axis synchronization.	Grinding
....		
....		
N31 Myy S300 ;	Rotates the spindle.	
N32 G01 G91 X... Y... F... ;	Makes movements for grinding.	
....		
....		
N40 Mzz ;	Stops the spindle.	
N41 G80.5 C0 ;	Cancels C-axis synchronization.	

V-axis synchronization mode (N10 to N21)

C-axis synchronization mode (N03 to N32)

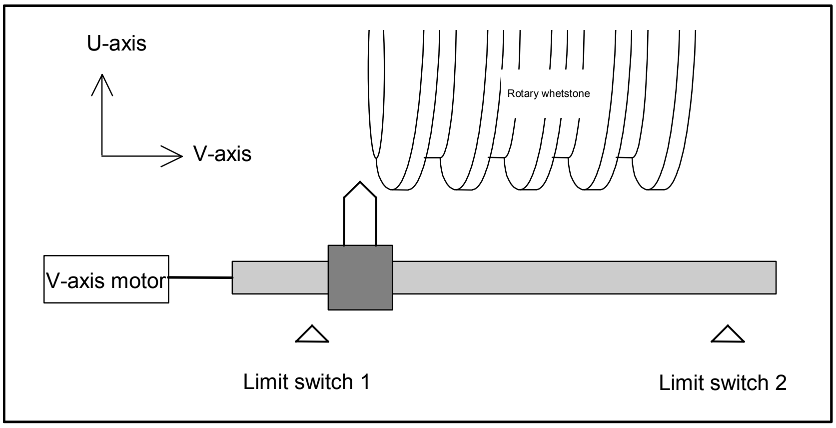
Thus, the synchronizations of two groups can be started and canceled independently of each other.

NOTE

If the V-axis (linear axis) is synchronized with the spindle as in dressing, the V-axis travel range is determined by the rotation of the spindle. To perform dressing with the tool moving back and forth along the V-axis in a certain range, therefore, the PMC must perform an operation in which the tool is stopped temporarily and is reversed when it reaches a certain position on the V-axis.
In the above example, limit switches are provided to determine the range of travel along the V-axis and the PMC performs control so that the whetstone rotates until the tool reaches the position of each limit switch on the V-axis.
By using the position switch function instead of limit switches, dressing can be performed as in the following example, without the need to mount limit switches to the machine. By rewriting the operating ranges of the position switches (parameters Nos. 6930 to 6945 and Nos. 6950 to 6965) using the G10 programming Parameter input, the range of travel along the V-axis can be specified using a program.

- Example of use of dressing

Gear grinder in the following machine configuration



O9500 ;	
N01 G01 G91 U__ F100 ;	Dressing axis approach
N02 Maa S100 ;	The Maa command causes the PMC to rotate the whetstone in the positive direction. In accordance with this, the tool moves along the V-axis in the + direction. When the tool reaches the position of limit switch 2 on the V-axis, the PMC stops the whetstone and returns FIN.
N02 U__ V__ ;	Movement to the next dressing position
N03 Mbb S100 ;	The Mbb command causes the PMC to rotate the whetstone in the negative direction. In accordance with this, the tool moves along the V-axis in the - direction. When the tool reaches the position of limit switch 1 on the V-axis, the PMC stops the whetstone and returns FIN.
N04 U__ V__ ;	Movement to the next dressing position If required, N02 to N04 are repeated to conduct dressing.
....	
....	
M99 ;	

- Command specification for hobbing machines

Based on the controlled axis configuration described in Fig. 20.6.4.3 (a), the sample program below sets the C-axis (in parameter 7710) for starting synchronization with the spindle according to the command specification method for hobbing machines.

O1234 ;	
N0010 M19 ;	Performs tool axis orientation.
N0020 G28 G91 C0 ;	Performs reference position return along the workpiece axis.
N0030 G81 T20 L1 ;	Starts synchronization with the spindle and C-axis at the ratio of a 18° rotation about the C-axis to one spindle rotation.
N0040 S300 M03 ;	Rotates the spindle at 300 min^{-1}.
N0050 G01 X... F... ;	Makes a movement along the X-axis (for cutting).
N0060 G01 Y... F... ;	Makes a movement along the Y-axis (for grinding). Axes such as the C-axis, X-axis, and Y-axis can be specified as required.
N0100 G01 X... F... ;	Makes a movement along the X-axis (for retraction).
N0110 M05 ;	Stops the spindle.
N0120 G80 ;	Cancels the synchronization between the spindle and C-axis.
N0130 M30 ;	

20.6.4.5 Synchronization ratio specification range

The programmed ratio (synchronization ratio) of a movement along the slave axis to a movement along the master axis is converted to a detection unit ratio inside the NC. If such converted data (detection unit ratio) exceeds a certain allowable data range in the NC, synchronization cannot be established correctly, and an alarm (PS1596) is issued.
Even when a programmed master axis movement and a programmed slave axis movement are within specifiable ranges, a detection unit ratio obtained by conversion can exceed the allowable range, thus resulting in an alarm.
Let K be a synchronization ratio. The internal data corresponding K is the amount of slave axis movement (Kn) represented in the detection unit divided by the amount of master axis movement (Kd) represented in the detection unit;
this fraction is represented as Kn/Kd (reduced to its lowest terms) as indicated below.

$$K = \frac{Kn}{Kd} = \frac{\text{Amount of slave axis movement represented in the detection unit}}{\text{Amount of master axis movement represented in the detection unit}}$$

Kn and Kd must lie within the following ranges:

$$-2147483648 \leq Kn \leq 2147483647$$
$$1 \leq Kd \leq 65535$$

When Kn or Kd exceeds its allowable range above, an alarm (PS1596) is issued.
In conversion to the detection unit, when the CMR (command multiplication:
parameter 1820) is a fraction or when inch/millimeter conversion is used, the fraction is directly converted without modification so that no error can occur in the conversion of specified amounts of movement.
During conversion, the amount of movement is multiplied by 254/100 for inch input on a millimeter machine, and 100/254 for metric input on an inch machine. Thus, Kn and Kd can become large numbers. If a synchronization ratio cannot be reduced to its lowest terms, an alarm condition is likely to occur.

- Example 1)

Based on the controlled axis configuration described in Fig. 20.6.4.3 (a), suppose that the spindle and V-axis are as follows:

Spindle pulse coder	:	72000pulse/rev (4 pulses for one A/B phase cycle)
C-axis least command increment	:	0.001deg
C-axis CMR	:	5
V-axis least command increment	:	0.001mm
V-axis CMR	:	5

Then, the C-axis detection unit is 0.0002 degree. The V-axis detection unit is 0.0002 mm. In this case, the synchronization ratio (Kn, Kd) is related with a command as indicated below. Here, let Pm and Ps be the amounts of movements represented in the detection unit on the master axis and slave axis specified in a synchronization start command, respectively.

(1) When the master axis is the spindle, and the slave axis is the C-axis

(a) Command : G81.5 T10 C0 L1 ;

Operation : Synchronization between the spindle and C-axis is started at the ratio of one rotation about the C-axis to ten spindle rotations.

Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10

Ps : (Amount of movement per rotation about the C-axis) × CMR × (one rotation) → 360000 × 5 × 1

$$\frac{Kn}{Kd} = \frac{360000 \times 5 \times 1}{72000 \times 10} = \frac{5}{2}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(b) Command : G81.5 T10 C0 L-1 ;

Operation : Synchronization between the spindle and C-axis is started at the ratio of one rotation about the C-axis to ten spindle rotations.
In this case, however, the direction of rotation is opposite to that of (a) above.

Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10

Ps : (Amount of movement per rotation about the C-axis) × CMR × (one rotation) → -360000 × 5 × 1

$$\frac{Kn}{Kd} = \frac{-360000 \times 5 \times 1}{72000 \times 10} = \frac{-5}{2}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(c) Command : G81.5 T1 C3.263 ;

Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis to one spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of C-axis movement) × CMR → 3263 × 5

$$\frac{Kn}{Kd} = \frac{3263 \times 5}{72000 \times 1} = \frac{3263}{14400}$$

Both Kn and Kd are within the allowable range. No alarm is output.

In this sample program, when T1 is specified for the master axis, the synchronization ratio (fraction) of the CMR of the C-axis to the denominator Kd can always be reduced to lowest terms, thus Kd falls in the allowable range. So, the specifiable range of C is as follows:

$-99999999 \leq C \leq 99999999$

(d) Command : G81.5 T10 C3.263 ;

Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis to ten spindle rotations.

Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10

Ps : (Amount of C-axis movement) × CMR → 3263 × 5

$$\frac{Kn}{Kd} = \frac{3263 \times 5}{72000 \times 10} = \frac{3263}{144000}$$

In this case, an alarm is issued because Kd exceeds the specifiable range.

(e) Command : G81.5 P10000 C-0.214 ;

Operation : Synchronization between the spindle and C-axis is started at the ratio of a -0.214 degree rotation of the C-axis to 10,000 feedback pulses from the pulse coder of the spindle.

Pm : (Specified number of feedback pulses from the pulse coder of the spindle) → 10000

Ps : (Amount of C-axis movement) × CMR → -214 × 5

$$\frac{Kn}{Kd} = \frac{-214 \times 5}{10000} = \frac{-107}{1000}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(2) When the master axis is the spindle, the slave axis is the V-axis (linear axis), and inch/metric conversion is performed

(a) For a millimeter machine and metric input

Command : G81.5 T1 V1.0 ;

Operation : Synchronization between the spindle and V-axis is started at the ratio of a 1.0 inch movement (25.4 mm) along the V-axis per spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of V-axis movement) × CMR → 1000 × 5

$$\frac{Kn}{Kd} = \frac{1000 \times 5}{72000} = \frac{5}{72}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(b) For a millimeter machine and inch input

Command : G81.5 T1 V1.0 ;

Operation : Synchronization between the spindle and V-axis is started at the ratio of a 1.0inch (25.4mm) movement along the V-axis per spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of V-axis movement) × CMR × 254 ÷ 100 → 10000 × 5 × 254 ÷ 100

$$\frac{Kn}{Kd} = \frac{10000 \times 5 \times 254}{72000 \times 100} = \frac{127}{72}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(c) For a millimeter machine and inch input

Command : G81.5 T1 V0.0013 ;

Operation : Synchronization between the spindle and V-axis is started at the ratio of a 0.0013 inch (0.03302 mm) movement along the V-axis per spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of V-axis movement) × CMR × 254 ÷ 100 → 13 × 5 × 254 ÷ 100

$$\frac{Kn}{Kd} = \frac{13 \times 5 \times 254}{72000 \times 100} = \frac{1651}{720000}$$

In this case, an alarm is issued because Kd exceeds the specifiable range.

- Example 2)

Based on the controlled axis configuration described in Fig. 20.6.4.3 (a), suppose that the spindle and V-axis are as follows:

Spindle pulse coder	:	72000pulse/rev (4 pulses for one A/B phase cycle)
C-axis least command increment	:	0.001deg
C-axis CMR	:	1/2
V-axis least command increment	:	0.001mm
V-axis CMR	:	1/2

Then, the C-axis detection unit is 0.002 degree. The V-axis detection unit is 0.002 mm.

In this case, the synchronization ratio (Kn, Kd) is related with a command as indicated below. Here, let Pm and Ps be the amounts of movements represented in the detection unit for the master axis and slave axis specified in a synchronization start command, respectively.

(1) When the master axis is the spindle, and the slave axis is the C-axis

(a) Command : G81.5 T1 C3.263 ;

Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis per spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of C-axis movement) × CMR → 3263 × 1 ÷ 2

$$\frac{Kn}{Kd} = \frac{3263 \times 1}{72000 \times 2} = \frac{3263}{144000}$$

In this case, an alarm is issued because Kd exceeds the specifiable range.

(b) Command : G81.5 T1 C3.26 ;

The value specified for C slightly differs from that specified for C in (a).

Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.26-degree rotation about the C-axis per spindle rotation.

Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1

Ps : (Amount of C-axis movement) × CMR → 3260 × 1 ÷ 2

$$\frac{Kn}{Kd} = \frac{3260 \times 1}{72000 \times 2} = \frac{163}{7200}$$

(a) causes an alarm to be output because the values cannot be abbreviated. (b) causes no alarm because the ratio of the travel distances can be abbreviated to a simple ratio.

20.6.4.6 Retract function

See "Retract function" in the Subsection 20.6.1 "Electronic Gear Box".

21 5-AXIS MACHINING FUNCTION

21.1 TOOL CENTER POINT CONTROL FOR 5-AXIS MACHINING

Overview

On a 5-axis machine having two rotary axes that turn a tool or table, this function performs tool length compensation constantly, even in the middle of a block, and exerts control so that the tool center point moves along the specified path. (See Fig. 21.1 (a).)
There are three different types of 5-axis machines - <1> those that rotate the tool only, <2> those that rotate the table only, and <3> those that rotate both the tool and table. (See Fig. 21.1 (d).)
This function is intended to perform machining on such 5-axis machines having rotary axes that turn a tool or table as well as three orthogonal axes (X-, Y-, and Z-axes) by accomplishing tool length compensation while changing the attitude of the tool. It enables the tool center point to move along the specified path even if the tool's direction changes with respect to the workpiece.
A coordinate system used for programming the tool center point control is called the programming coordinate system.
A coordinate system fixed on the table can be used as the programming coordinate system, which makes CAM programming easy.
A workpiece coordinate system fixed on a machine coordinate system can be employed as the programming coordinate system as well. On a machine of mixed type or table rotation type, cutter compensation for 5-axis machining is programmed in the workpiece coordinate system. So, to use cutter compensation for 5-axis machining and tool center point control at the same time, the workpiece coordinate system must be used as the programming coordinate system.
In any case, the cutting speed can be controlled easily because the tool center point moves at a specified speed with respect to the table (workpiece).
The commands that can be issued during tool center point control are positioning (G00), linear interpolation (G01), circular interpolation (G02, G03), and helical interpolation (G02, G03).

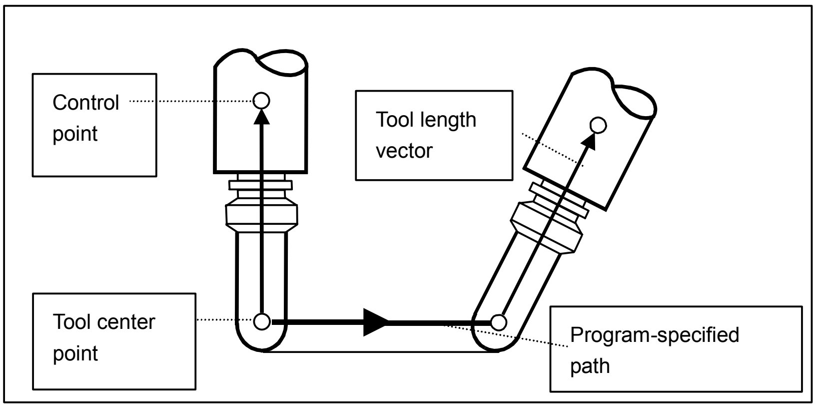


Fig. 21.1 (a) Path of the tool center point

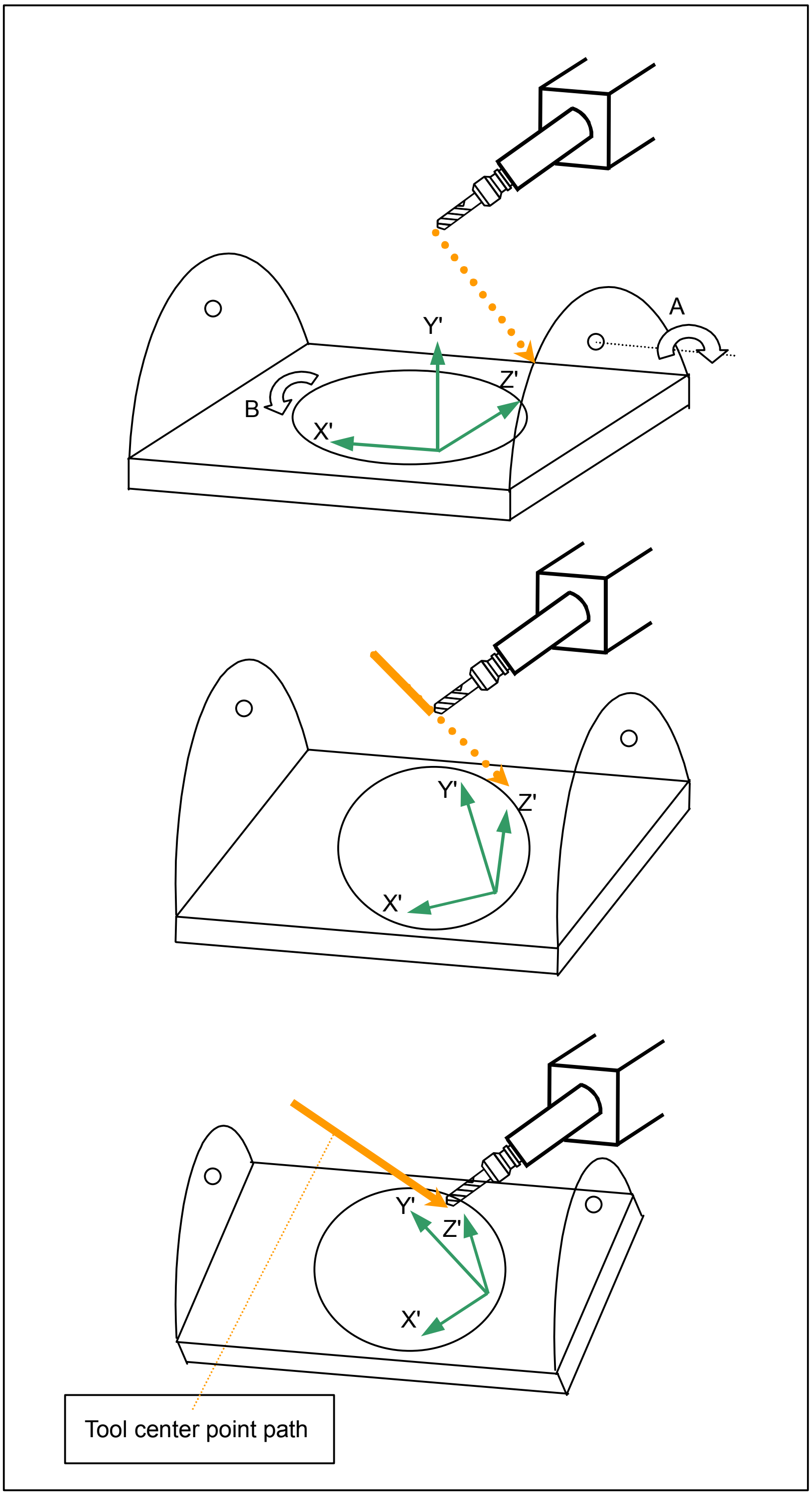


Fig. 21.1 (b) Path of the tool center point

When a coordinate system fixed on the table is used as the programming coordinate system, programming can be performed without worrying about the rotation of the table because the programming coordinate system does not move with respect to the table, although the position and direction of the workpiece fixed on the table change due to its rotation. When a straight line is specified, the tool center point moves along a straight path with respect to the workpiece as instructed. (See Fig. 21.1 (b).)
By setting the relevant parameter, the workpiece coordinate system can also be employed as the programming coordinate system. In this case, as the table turns, the position and direction of the workpiece fixed on the table change with respect to the programming coordinate system. It is therefore necessary to take into account the rotation of the table when specifying the end point. In this case, too, when a straight line is specified, the tool center point moves along a straight path with respect to the workpiece as instructed.
Fig. 21.1 (c) illustrates how linear interpolation is accomplished with a mixed type machine, showing the relationship between the case when a table-fixed coordinate system is used as the programming coordinate system and the case when the workpiece coordinate system is used.
If linear interpolation is specified in this function mode, speed control is exerted in such a way that the tool center point moves at a specified speed with respect to the workpiece.

Example)

Machine configuration: The A-axis is the rotation axis for controlling the tool.
The B-axis is the rotation axis for controlling the table.

Program: Created using the programming coordinate system.

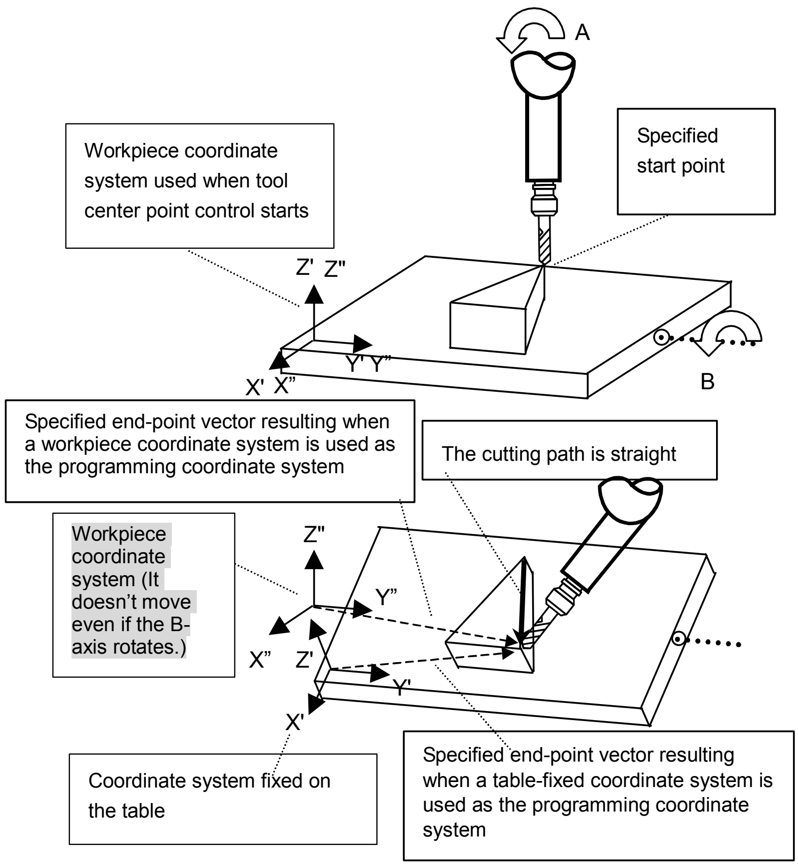


Fig. 21.1 (c) Linear interpolation with a mixed type machine

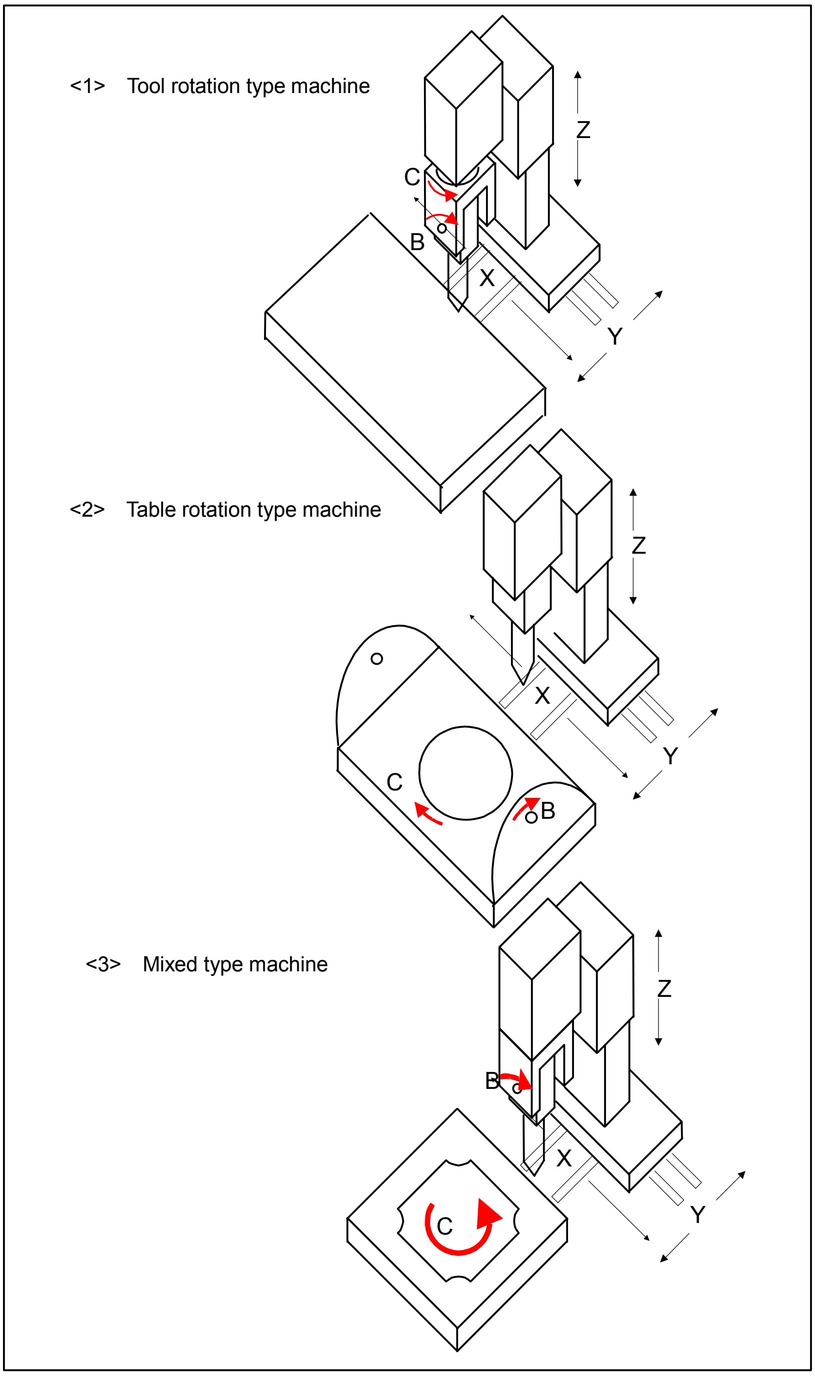


Fig. 21.1 (d) Three types of 5-axis machine

Even if the rotary axis that controls the tool does not intersect the one that controls the table, this function can still be used.

There are two types, as described below, one of which is used depending on how the direction of the tool axis is specified.

(1) Type 1
The block end point of the rotary axes is specified (e.g. A, B, C).
The CNC performs tool length compensation by the specified amount in the tool axis direction that is calculated from the specified position of the rotary axes and exerts control so that the tip of the tool moves along the specified path.

(2) Type 2
The direction of the tool axis (I, J, K) at the block end point, as seen from the coordinate system fixed on the table, is specified, instead of the position of the rotary axes.
The CNC calculates an end point of the rotary axes where the tool will face the specified direction, performs tool length compensation by the specified amount in the tool axis direction that is calculated from the position of the rotary axes, and exerts control so that the tip of the tool moves along the specified path.

Format

- Positioning and linear interpolation for tool center point control (type 1)

G43.4 IP_ α_ β_ H_ ; Starts tool center point control (type 1).
IP_ α_ β_ ;
:

IP : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement

α, β : In the case of an absolute programming, the coordinate value of the end point of the rotary axes
In the case of an incremental programming, the amount of the rotary axis movement

H : Tool offset number

Movement to the position specified by the G43.4 block (startup) does not constitute tool center point control. Only tool length compensation is performed.
As for rotary axes, either table rotation axes or tool rotation axes are specified.

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves along a straight line with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

- Positioning and linear interpolation for tool center point control (type 2)

G43.5 IP_ H_ Q_ ; Starts tool center point control (type 2).
IP_ I_ J_ K_ ;
:

IP	: In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement
I, J, K	: Tool axis direction at the block end point as seen from the programming coordinate system
H	: Tool offset number
Q	: Inclination angle of the tool (in degrees)

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.
No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).
With a tool rotation type machine, I, J, K can be specified using the G43.5 block. In the case of a table rotation type or mixed type machine, however, these cannot be specified. Specifying them with a table rotation type or mixed type machine causes alarm PS5421.

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves along a straight line with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ **CAUTION**

1 If one or two of the I, J, and K values are omitted, the omitted value or values are considered to be 0.
2 In a block in which I, J, and K are all omitted, the compensation vector of the preceding block is used.
3 This block can be used only when the programming coordinate system is fixed on the table (when the WKP parameter (No.19696#5) is set to 0). Specifying G43.5 when the WKP parameter (No.19696#5) is set to 1 causes alarm PS5459.
4 Type 2 cannot be used when there is only one rotary axis or when any hypothetical axis is used. Specifying G43.5 in such a case causes alarm PS5459.
5 When using the rotary axis rollover function or the rotary axis control function, set parameter No.1260 (amount of movement per rotation of the rotary axis) to 360 degrees.

- Circular interpolation for tool center point control (type 1)

G43.4 IP_ H_ ; Starts tool center point control (type 1).

$$\mathbf{G17}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ IP\_\left\{\begin{matrix}\mathbf{I\_J\_K}\\ \mathbf{R\_}\end{matrix}\right\}\ \alpha\_\beta\_\mathbf{F\_};$$

$$\mathbf{G18}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ IP\_\left\{\begin{matrix}\mathbf{I\_J\_K\_}\\ \mathbf{R\_}\end{matrix}\right\}\ \alpha\_\beta\_\mathbf{F\_};$$

$$\mathbf{G19}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ IP\_\left\{\begin{matrix}\mathbf{I\_J\_K}\\ \mathbf{R\_}\end{matrix}\right\}\ \alpha\_\beta\_\mathbf{F\_};$$

:

G17	:	X-Y plane of the table coordinate system
G18	:	Z-X plane of the table coordinate system
G19	:	Y-Z plane of the table coordinate system
G02	:	Clockwise (CW) circular interpolation
G03	:	Counterclockwise (CCW) circular interpolation
IP	:	In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)
I, J, K	:	Specify the distance between the start point in the rotary axis position of the block start point and the center of the arc, as seen from the programming coordinate system.
R	:	Arc radius R > 0: The center angle of the arc is less than 180°. R < 0: The center angle of the arc is more than 180°.
α, β	:	In the case of an absolute programming, the coordinate value of the end point of the rotary axis In the case of an incremental programming, the amount of the rotary axis movement
F	:	Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)
H	:	Tool offset number

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed. While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves along an arc with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ **CAUTION**

Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.

- Circular interpolation for tool center point control (type 2)

G43.5 IP_ H_ Q_ ; Starts tool center point control (type 2).

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} \text{IP\_ I\_ J\_ K\_ R\_ F\_;}$$

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} \text{IP\_ I\_ J\_ K\_ R\_ F\_;}$$

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} \text{IP\_ I\_ J\_ K\_ R\_ F\_;}$$

:

G17	: X-Y plane of the table coordinate system
G18	: Z-X plane of the table coordinate system
G19	: Y-Z plane of the table coordinate system
G02	: Clockwise (CW) circular interpolation
G03	: Counterclockwise (CCW) circular interpolation
IP	: In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)
I, J, K	: Tool axis direction at the block end point as seen from the programming coordinate system
R	: Arc radius R > 0: The center angle of the arc is less than 180°. R < 0: The center angle of the arc is more than 180°.
F	: Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)
H	: Tool offset number
Q	: Inclination angle of the tool (in degrees)

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.

No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).

With a tool rotation type machine, I, J, K can be specified using the G43.5 block. In the case of a table rotation type or mixed type machine, however, these cannot be specified. Specifying them with a table rotation type or mixed type machine causes alarm PS5421.

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves along an arc with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ **CAUTION**

1. Only arc radius R can be specified. (The distance from the start point to the center of the arc cannot be specified using I, J, and K.)
2. A round circle (the start point and end point are the same) cannot be specified.
   Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.
3. See the CAUTION box for tool center point control (type 2).

- Helical interpolation for tool center point control (type 1)

G43.4 IP_ H ; Starts tool center point control (type 1).

$$\mathbf{G17}\left\{\begin{array}{c}\mathbf{G02}\\ \mathbf{G03}\end{array}\right\}\mathrm{IP}\_\left\{\begin{array}{c}\mathbf{I\_J\_K\_}\\ \mathbf{R\_}\end{array}\right\}\alpha\_\beta\_\gamma\_\mathbf{F\_};$$

$$\mathbf{G18}\left\{\begin{array}{c}\mathbf{G02}\\ \mathbf{G03}\end{array}\right\}\mathrm{IP}\_\left\{\begin{array}{c}\mathbf{I\_J\_K\_}\\ \mathbf{R\_}\end{array}\right\}\alpha\_\beta\_\gamma\_\mathbf{F\_};$$

$$\mathbf{G19}\left\{\begin{array}{c}\mathbf{G02}\\ \mathbf{G03}\end{array}\right\}\mathrm{IP}\_\left\{\begin{array}{c}\mathbf{I\_J\_K\_}\\ \mathbf{R\_}\end{array}\right\}\alpha\_\beta\_\gamma\_\mathbf{F\_};$$

:

G17	: X-Y plane of the table coordinate system
G18	: Z-X plane of the table coordinate system
G19	: Y-Z plane of the table coordinate system
G02	: Clockwise (CW) circular interpolation
G03	: Counterclockwise (CCW) circular interpolation
IP	: In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)
I, J, K	: Specify the distance between the start point in the rotary axis position of the block start point and the center of the arc, as seen from the programming coordinate system.
R	: Arc radius R > 0: The center angle of the arc is less than 180°. R < 0: The center angle of the arc is more than 180°.
α, β	: In the case of an absolute programming, the coordinate value of the end point of the rotary axis In the case of an incremental programming, the amount of the rotary axis movement
γ	: In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement (This pertains only to one of the linear axes subject to tool center point control for 5-axis machining which does not exist on the plane. Linear interpolation is performed simultaneously with circular interpolation.)
F	: Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)
H	: Tool offset number

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.

Because the specified speed is usually the speed in the tangent direction of the arc, the speed of the linear axis, when seen from the table coordinate system, is: $F \times \dfrac{\text{Length of the linear axis}}{\text{Length of the arc}}$.

Depending on parameter HTG (No.1403#5), the specified speed varies as shown in the following table.

HTG (No.1403#5)	
0	1
Tangential speed of the arc	Synthetic speed of the linear axis speed and tangential speed

While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves helically with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ **CAUTION**

Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.

- Helical interpolation for tool center point control (type 2)

G43.5 IP_ H_ Q_; Starts tool center point control (type 2).

$$\text{G17}\left\{\begin{matrix}\text{G02}\\ \text{G03}\end{matrix}\right\}\ \text{IP\_ I\_ J\_ K\_ R\_}\ \gamma\text{\_ F\_;}$$

$$\text{G18}\left\{\begin{matrix}\text{G02}\\ \text{G03}\end{matrix}\right\}\ \text{IP\_ I\_ J\_ K\_ R\_}\ \gamma\text{\_ F\_;}$$

$$\text{G19}\left\{\begin{matrix}\text{G02}\\ \text{G03}\end{matrix}\right\}\ \text{IP\_ I\_ J\_ K\_ R\_}\ \gamma\text{\_ F\_;}$$

:

G17	:	X-Y plane of the table coordinate system
G18	:	Z-X plane of the table coordinate system
G19	:	Y-Z plane of the table coordinate system
G02	:	Clockwise (CW) circular interpolation
G03	:	Counterclockwise (CCW) circular interpolation
IP	:	In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)
I, J, K	:	Tool axis direction at the block end point as seen from the programming coordinate system
R	:	Arc radius R > 0: The center angle of the arc is less than 180°. R < 0: The center angle of the arc is more than 180°.
γ	:	In the case of an absolute programming, the coordinate value of the end point of the tool tip movement In the case of an incremental programming, the amount of the tool tip movement (This pertains only to one of the linear axes subject to tool center point control for 5-axis machining which does not exist on the plane. Linear interpolation is performed simultaneously with circular interpolation.)
F	:	Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)
H	:	Tool offset number
Q	:	Inclination angle of the tool (in degrees)

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.

Because the specified speed is the speed in the tangent direction of the arc, the speed of the linear axis, when seen from the table coordinate system, is: $F \times \dfrac{\text{Length of the linear axis}}{\text{Length of the arc}}$.

Depending on parameter HTG (No.1403#5), the specified speed varies as shown in the following table.

HTG (No.1403#5)	
0	1
Tangential speed of the arc	Synthetic speed of the linear axis speed and tangential speed

No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).
With a tool rotation type machine, I, J, K can be specified using the G43.5 block. In the case of a table rotation type or mixed type machine, however, these cannot be specified. Specifying them with a table rotation type or mixed type machine causes alarm PS5421.
While performing compensation for the rotary axes, the CNC controls the control points so that the tool center point moves helically with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ CAUTION

1 Only arc radius R can be specified. (The distance from the start point to the center of the arc cannot be specified using I, J, and K.)
2 A round circle cannot be specified.
3 Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.
4 See the CAUTION box for tool center point control (type 2).

- Tool center point control cancellation command

G49 IP_ α_ β_ ; Cancels tool center point control.

IP : In the case of an absolute programming, the coordinate value of the end point of the tool control point movement
In the case of an incremental programming, the amount of the tool control point movement

α, β : In the case of an absolute programming, the coordinate value of the end point of the rotary axes
In the case of an incremental programming, the amount of the rotary axis movement

The cancellation block for tool center point control is the one that controls buffering.

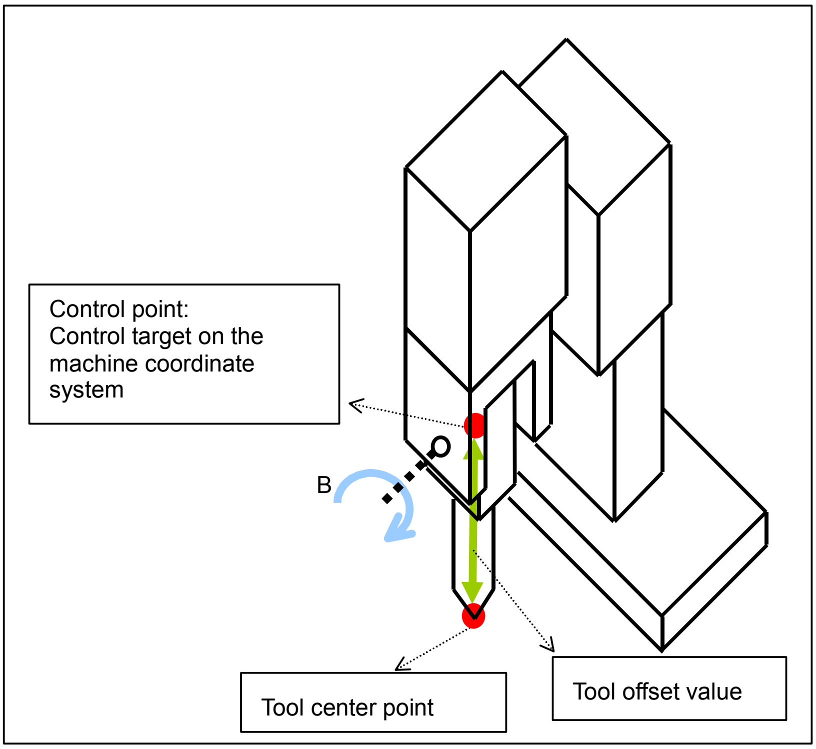
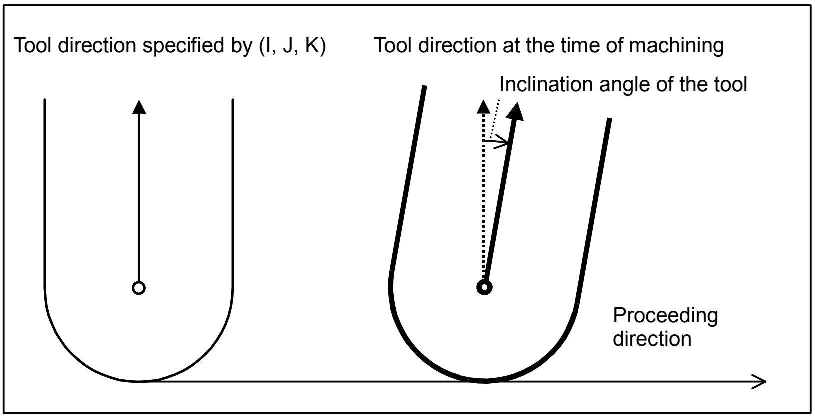


Fig. 21.1 (e) Control point and tool center point

> ⚠ **CAUTION**
> The G49 command must be executed in G00 or G01 mode.

- Inclination angle of the tool

In the case of tool center point control of type 2, the inclination angle of the tool can be specified using address Q of G43.5. The inclination angle of the tool represents how inclined the tool direction is toward the proceeding direction with respect to the direction specified by (I, J, K) at the time of machining on the plane produced by the tool direction specified by (I, J, K) and the proceeding direction on the programming coordinate system. (See the figure below.)
Overviewly, the normal direction on the machining plane is specified by (I, J, K). If the tool direction needs to be inclined toward the proceeding direction with respect to the normal direction at the time of machining, perform compensation using Q.
If the direction specified by (I, J, K) matches the direction in which the tool moves at the time of machining, commanding Q is unnecessary.



Example)

To incline the tool by two degrees toward the proceeding direction at the time of machining, enter the following command:
G43.5 I_ J_ K_ H_ Q2.0

Explanation

- When a coordinate system fixed on the table is used as the programming coordinate system

The programming coordinate system is used for tool center point control. When the G43.4 or G43.5 command is specified with parameter WKP (No.19696#5) set to 0, the workpiece coordinate system that is fixed on the table at that point of time becomes the programming coordinate system. Thereafter, the programming coordinate system rotates as the table turns around.
It does not rotate with the tool head.
X, Y, and Z mentioned hereinafter are assumed to be commanded on the programming coordinate system.
When the G43.4 or G43.5 command is specified or when the rotary axis of the table has moved in a block preceding G43.4 or G43.5, the angle of the table's rotary axis represents the initial state of the programming coordinate system.
In the case of type 2, the tool direction seen from the coordinate system that is fixed on the table is specified by I, J, K.
In the descriptions that follow, the table-fixed coordinate system is represented by X', Y', and Z'.

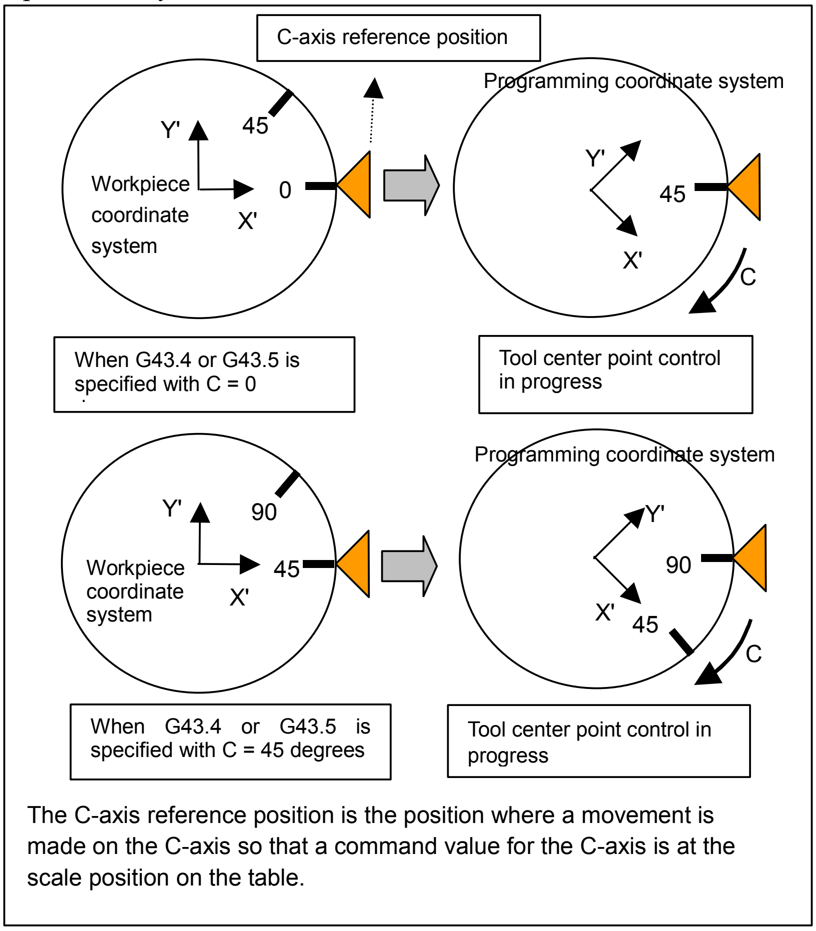


Fig. 21.1 (f) Programming coordinate system fixed on the table

- When the workpiece coordinate system is used as the programming coordinate system

When the G43.4 command is specified with parameter WKP (No.19696#5) set to 1, the workpiece coordinate system that is in use at that point of time becomes the programming coordinate system.
In this case, the programming coordinate system does not rotate as the table turns around but remains fixed on the workpiece coordinate system.
Hereinafter, when X, Y, and Z are commanded, the tool moves along a straight line toward the table (workpiece). For X, Y, Z, specify the end point location after the rotation of the table, as seen from the programming coordinate system.
Type 2 cannot be used. Specifying G43.5 when parameter WKP (No.19696#5) set to 1 causes alarm PS5459.
In the descriptions that follow, the coordinate values of the workpiece coordinate system used as the programming coordinate system are represented by X", Y", and Z".

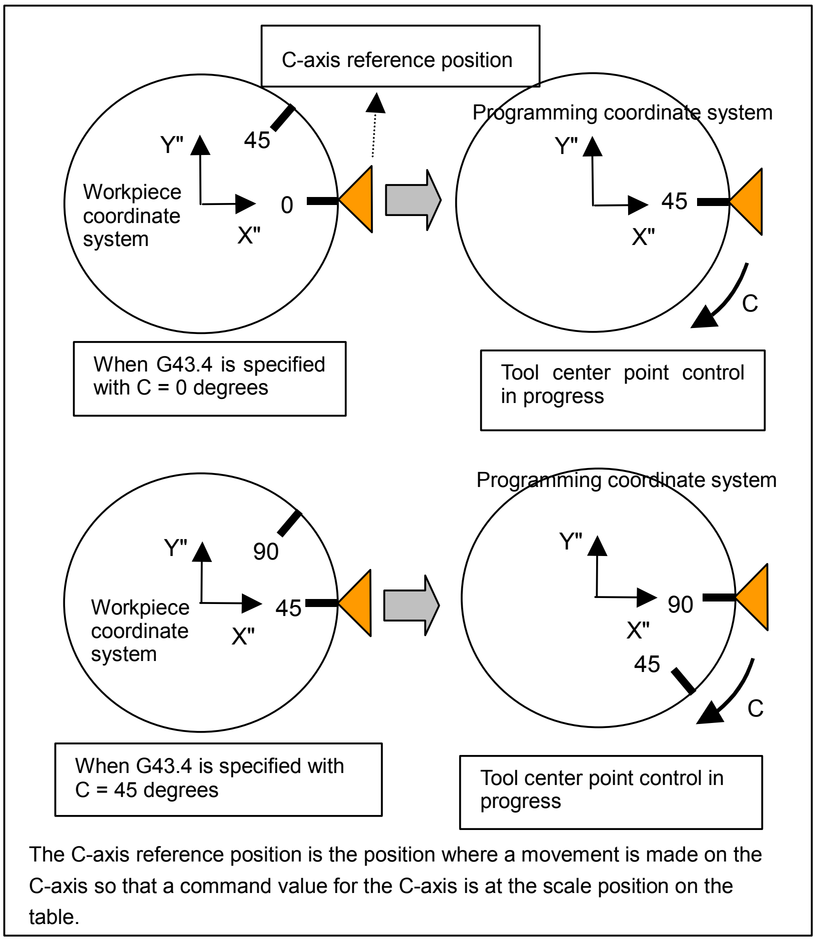


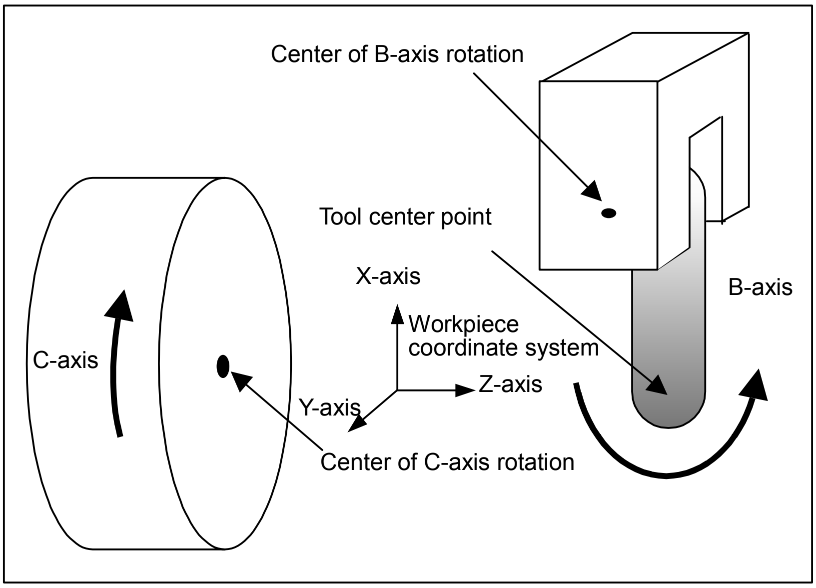
Fig. 21.1 (g) Programming coordinate system identical to the workpiece coordinate system

- Notes on performing circular interpolation and helical interpolation when using the workpiece coordinate system as the programming coordinate system

- The start point, end point, and center of an arc change as the table rotation axis rotates.
- I, J, K commands the vector of the block start point to the center of the arc from the start point in the rotary axis position.
- Note the following:
  <1> Only a table rotation axis normal to a selected plane can be rotated during circular interpolation.
  <2> During circular interpolation, those table rotation axes not normal to a selected plane must continue to be at the same position as when tool center point control is started.
  If <1> or <2> is not satisfied, the alarm PS5421 is issued.
  No restriction is imposed on rotation on a tool rotation axis.

- Program examples

- In the case of a mixed type machine
  Descriptions are based on the following machine configuration.



When the G17 (X-Y plane) command is executed

After the G43.4 command, the X-Y plane is selected using the G17 command and circular interpolation is performed by rotating the C-axis (table rotation axis) (including those cases where the C-axis moves before the G43.4 command). → This case corresponds to <1> and allows circular interpolation.

Example)

```
 :
(G01 C90. ;)
G43.4 H1 ;
G17 G02 IP IR   B10. C20. ;
 :
```

IP: Coordinates of the end point

IR: Arc radius

When the G18 (Z-X plane) or G19 (Y-Z plane) command is executed

After the G43.4 command, the Z-X plane is selected using the G18 command and circular interpolation is performed without rotating the C-axis (including those cases where the C-axis moves before the G43.4 command). → This case corresponds to <2> and allows circular interpolation.

The same is also true when the G19 command is used.

Example)

```
 :
G43.4 H1 ;
G18 G02 IP IR C20. ;
 :
```

After the G43.4 command, the Z-X plane is selected using the G18 command and the C-axis is rotated during circular interpolation . → Alarm (violation of <2>)

The same is also true when the G19 command is used.

Example)

```
 :
G43.4 H1 ;
G18 G02 IP IR C20. ;
 :
```

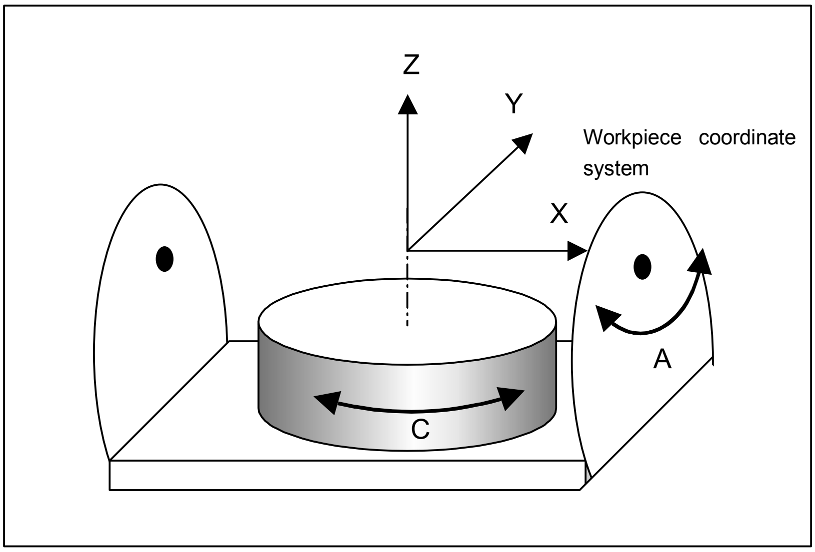
After the G43.4 command, the Z-X plane is selected using the G18 command and circular interpolation is performed after rotating the C-axis. → Alarm (violation of <2>)

The same is also true when the G19 command is used.

Example)

```
 :
G43.4 H1 ;
G01 C10.
G18 G02 IP IR ;
 :
```

- In the case of a table rotation type machine
  Descriptions are based on the following machine configuration.
  A table rotation type machine can be considered equivalent to a mixed type machine if any of its two table rotation axes does not move.



When the G17 (X-Y plane) command is executed
The master axis (A-axis) moves before the G43.4 command and, after the G43.4 command, circular interpolation is performed without rotating the A- or C-axis (including those cases where the C-axis moves before the G43.4 command). → This case corresponds to <2> and allows circular interpolation.
Example)

```
 :
G01 A90. (C10.);
G43.4 H1 ;
G17 G02 IP IR ;
 :
```

The master axis (A-axis) moves before the G43.4 command and, after the G43.4 command, circular interpolation is performed using the G17 (X-Y plane) command by rotating the C-axis, or the C-axis is rotated during circular interpolation. → Alarm (violation of <2>)
Example)

```
  :
G01 A90.;
G43.4 H1 ;
G01 C10. ;
G17 G02 IP IR ;
  :
  :
G01 A90.;
G43.4 H1 ;
G17 G02 IP IR C10.;
  :
```

After the G43.4 command, the A-axis is moved and circular interpolation is performed using the G17 command (X-Y plane). → Alarm (violation of <2>)
Example)

```
  :
G43.4 H1 ;
G01 A10. ;
G17 G02 IP IR C10.;
  :
```

G19 (Y-Z plane) command is executed
The G43.4 command is executed after moving the A-axis and circular interpolation is performed while rotating the A-axis using the G19 (Y-Z plane) command. → This case corresponds to <1> and allows circular interpolation.
Example)

```
  :
G01 A90. ;
G43.4 H1 ;
G19 G02 IP IR A10. ;
  :
```

After the G43.4 command, the C-axis is rotated and circular interpolation is performed using the G19 (Y-Z plane) command. → Alarm (violation of <2>)
Example)

```
  :
G43.4 H1 ;
G01 C10. ;
G19 G02 IP IR ;
  :
```

When the G18 (Z-X plane) command is executed

The G43.4 command is executed after moving the A- and C-axes, and circular interpolation is performed using the G18 (Z-X plane) command without moving any rotary axis. → This case corresponds to <2> and allows circular interpolation.

Example)

```
 :
G01 A90. C10. ;
G43.4 H1 ;
G18 G02 IP IR;
 :
```

After the G43.4 command, circular interpolation is performed using the G18 command (Z-X plane) by moving any of the rotary axes. → Alarm (violation of <2>)

Example)

```
 :
G43.4 H1 ;
G01 A10. (C10.)
G18 G02 IP IR;
 :
```

- Tool center point control command

During tool center point control, the command specifies the location of each block end point as seen from the programming coordinate system.

The program specifies the tool center point.

As for the rotary axis, the command specifies the coordinate values of each block end point in the case of type 1 or the tool direction at each block end point in the case of type 2.

The feedrate is specified by the tangential speed relative to the workpiece (the tool's relative speed as opposed to the workpiece), which is represented by F.

- Commands that can be specified during tool center point control

The commands that can be specified during tool center point control are linear interpolation (G01), positioning (G00), circular interpolation (G02, G03), and helical interpolation (G02, G03).

When linear interpolation (G01) is specified during tool center point control, speed control is exerted so that the tool center point moves at the specified speed.

The circular interpolation command (G02, G03) controls the tangential speed of the arc path along which the tool center point moves.

The helical interpolation command (G02, G03) controls the tangential speed of the arc path along which the tool center point moves or a synthetic speed including that of the helical axis. (This is dependent on the setting of parameter HTG (No.1403#5).)

As the actual speed, the speed at the control point is shown.

NOTE

Tool center point control requires either the AI contour control I or AI contour control II option. In addition, be sure to specify the following parameters:

(1) Parameter LRP (No.1401#1)=1 :
Linear rapid traverse
(2) Parameter FRP (No.19501#5)=1 :
Acceleration/deceleration before interpolation is used for rapid traverse.
(3) Parameter (No.1671) :
Acceleration before interpolation for rapid traverse.
(4) Parameter (No.1672) :
Change time for bell-shaped acceleration before interpolation for rapid traverse.
(5) Parameter (No.1660) :
Maximum permissible acceleration for acceleration/deceleration before interpolation

If they are not specified, alarm PS5420 is generated.

- Rotary axis command

If a command is specified during tool center point control that prohibits the tool center point from moving with respect to the workpiece, the maximum cutting speed (parameter No.1430) is assumed as the feedrate of the rotary axis when parameter RFC (No.19696#6) is 0, and the speed specified by F is assumed when parameter RFC (No.19696#6) is 1.

The rotary axis command cannot be specified during tool center point control of type 2. Specifying the command with type 2 causes alarm PS5421.

- The moving distance of the rotary axis is long compared to that of the linear axis

If the moving distance of the rotary axis is long compared to that of the linear axis, the rotary axis moves faster so that the tool center point moves at the specified speed, possibly resulting in the tool center point traveling on an inadequate path.

In such a case, it is possible to exert control to slow down the speed and ensure that the tool center point travels on the specified path, by setting parameter CRS (No.19746#6) to 1. When using this control, specify in parameter (No.19751) (for rapid traverse) and parameter (No.19752) (for cutting feed) the extent of deviation of the path at which the speed is to be slowed down (maximum allowable deviation of the path). When 0 is specified, the least input increment is regarded as the maximum allowable deviation of the path.

- Tool behavior at startup and cancellation

When tool center point control is started (G43.4/G43.5) or canceled (G49), the tool moves by a tool offset value.
Compensation vector calculation is performed only at the end of a block.

- Current position display during tool center point control

During tool center point control, the position of the control point (rotation center of the tool rotation axis) is displayed as the machine coordinate.
When parameter WKP (No.19696#5) is 0, whether to use absolute or relative coordinates can be selected using parameter DET (No.19608#2).
If parameter DET (No.19608#2) is 0, the position of the tool center point on the programming coordinate system is displayed.
If parameter DET (No.19608#2) is 1, the position of the tool center point in the workpiece coordinate system is displayed.

- Tool offset

If tool offsets are used based on tool numbers, tool center point control is carried out using the tool length compensation value corresponding to the relevant tool number (T code).
If tool life management is used, tool center point control is carried out using the tool length compensation value corresponding to the tool in use.

- Angle of the rotary axis for type 2 (when the movement range is not specified)

When the direction of the tool is specified by I, J, K, Q for type 2, more than two pairs of "computed angles" of the rotary axes usually exist.
The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the specified tool axis direction.
The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.

The following descriptions assume that there is no movement range specification (parameter No.19741 - No.19744 = 0).

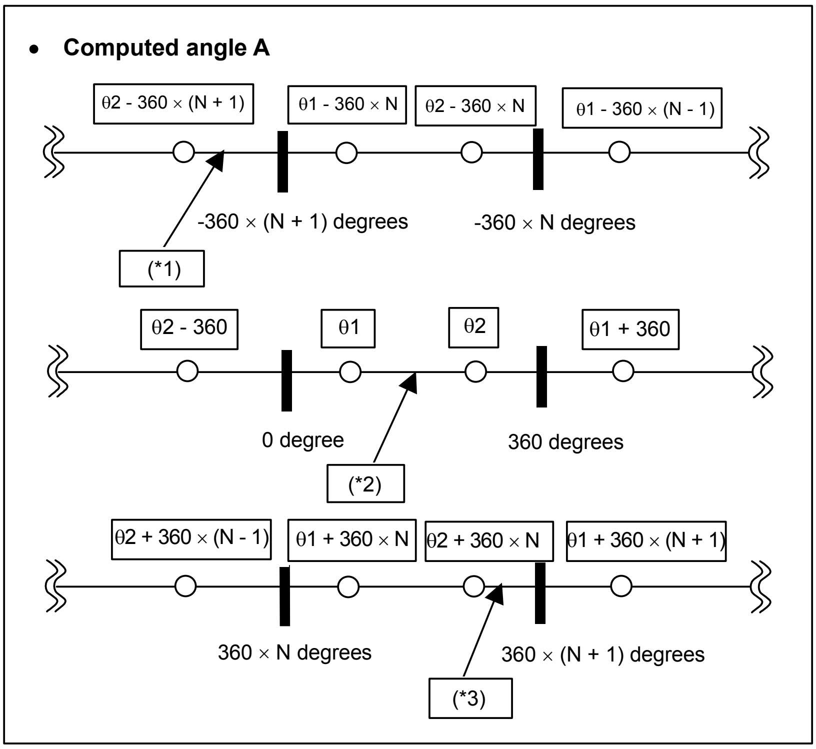
"Output judgment conditions"

Tool rotation type or table rotation type machine	
<1>	The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller.
	↓ ↓ When the master axis moving angle is the same ↓
<2>	The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller.
	↓ ↓ When the slave axis moving angle is the same ↓
<3>	The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
	↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4>	The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Mixed type machine	
<1>	The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller.
	↓ ↓ When the table moving angle is the same ↓
<2>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.
	↓ ↓ When the tool moving angle is the same ↓
<3>	The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
	↓ ↓ When the table angle is equally near to 0 degree ↓
<4>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

The process of judging whether the moving angle is smaller or larger as the output judgement condition is called "movement judgement."
When parameter PRI (No.19608#5) is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

The "movement judgement" process is explained below.
When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."
Usually, two pairs of "basic computed angles" exist.
For example, assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A θ1 degree; B ϕ1 degree)
(A θ2 degrees; B ϕ2 degrees) where θ1 ≤ θ2.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.
Based on the PA angle, the "movement judgement" process is done as follows (when parameter PRI (No.19608#5) is 0).



"Movement judgment"

When the PA angle is (*1):
The output angle is: (A $\theta 2$ - 360 × (N + 1) degrees; B $\phi 2$ degrees).
Namely, $\theta 2$ - 360 × (N + 1) degrees is adopted that is nearer to the computed angle of A, and $\phi 2$, which is the same group as $\theta 2$, is adopted as the computed angle of B.

When the PA angle is (*2):
The output angle is: (A $\theta 1$ degrees; B $\phi 1$ degrees).
Namely, $\theta 1$ degrees is adopted that is nearer to the computed angle of A, and $\phi 1$, which is the same group as $\theta 1$, is adopted as the computed angle of B.

When the PA angle is (*3):
The output angle is: (A θ + 360 × N degrees; B $\phi 2$ degrees).
Namely, $\theta 2$ + 360 × N degrees is adopted that is nearer to the computed angle of A, and $\phi 2$, which is the same group as $\theta 2$. is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgement" is made for rotary axis B (slave) according to the "output judgment conditions."
If the "output angle" of rotary axis A is determined by the "movement judgement" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis B.
Similarly, if the "output angle" of rotary axis B is determined by the "movement judgement" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

The "output angle" is explained below using a tool rotation type machine as an example.
This example illustrates a machine having a "BC type tool axis Z."

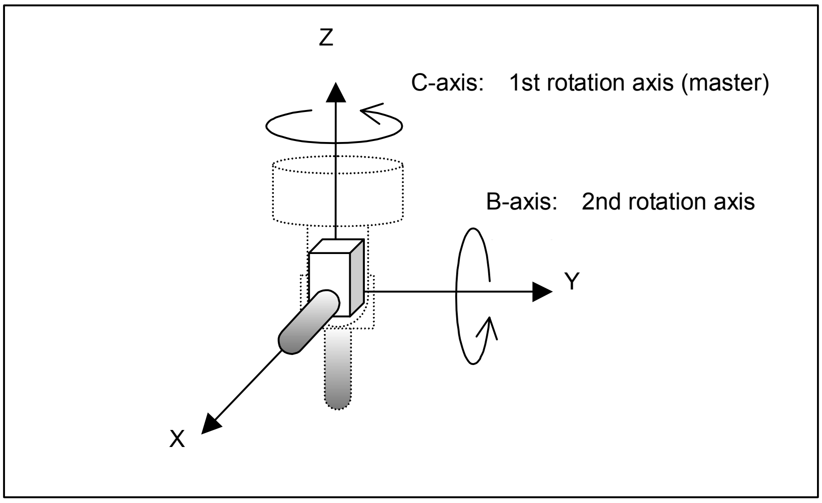


Fig. 21.1 (h) BC type tool axis Z

The following two pairs of "computed basic angles" exist that direct the tool axis toward the + X axis direction.
(B 90 degrees; C 180 degrees)
(B 270 degrees; C 0 degree)

<1> When the current rotary axis angles are (B -70 degrees; C 30 degrees)
The "output angles" are (B -90 degrees; C 0 degree).
0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).

<2> When the current rotary axis angles are (B 80 degrees; C 500 degrees)
The "output angles" are (B 90 degrees; C 540 degrees).
540 degrees (180 degrees + 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.

<3> When the current rotary axis angles are (B 60 degrees; C 90 degrees)
The "output angles" are (B 90 degrees; C 180 degrees).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.

<4> When the current rotary axis angles are (B 180 degrees; C 90 degrees)
The "output angles" are (B 270 degrees; C 0 degree).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree.
That is, the pair is adopted whose C axis angle is 0 degree and whose B axis angle is 270 degrees.

When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.
In that case, the master axis does not move from the current angle.

An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

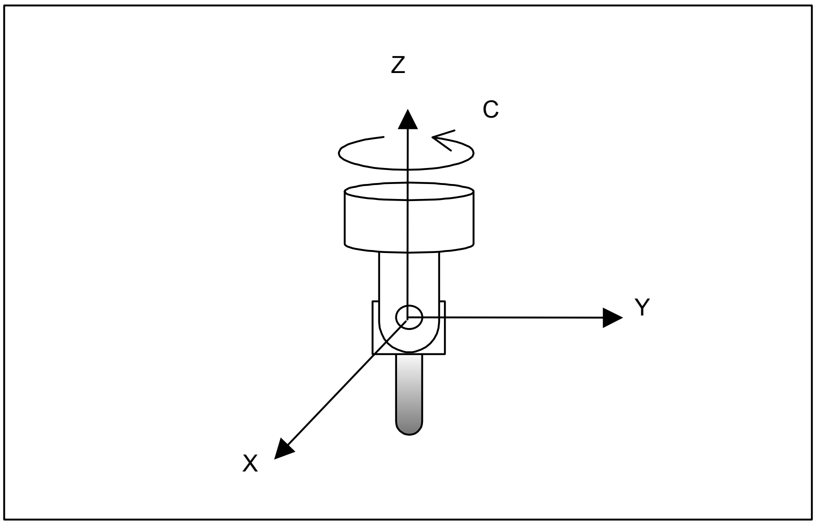


Fig. 21.1 (i) BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

- Angle of the rotary axis for type 2 (when the movement range is specified)

If the upper and lower limits of the movement range of the rotary axis is specified using parameters No.19741 to No.19744, the rotary axis will move only within the specified range when the direction is specified using I, J, K, Q command for type 2 control.

Although the procedure for determining the angles is the same as that used "when the movement range is not specified," the "output angles" need to be selected from **<u>those computed angles that are within the specified movement range for both axes</u>**.

"Output judgment conditions"

Tool rotation type or table rotation type machine

<1> Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller represents the "output angles."

↓
↓ When the master axis moving angle is the same
↓

<2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller.

↓
↓ When the slave axis moving angle is the same
↓

<3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

↓
↓ When the master axis angle is equally near to 0 degree
↓

<4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Mixed type machine

<1> Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose table (second rotary axis) moving angle is smaller represents the "output angles."

↓
↓ When the table moving angle is the same
↓

<2> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.

↓
↓ When the tool moving angle is the same
↓

<3> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

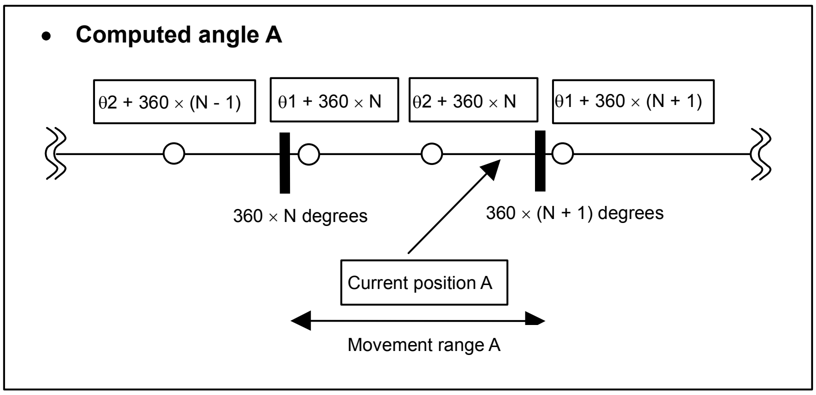
↓
↓ When the table angle is equally near to 0 degree
↓

<4> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

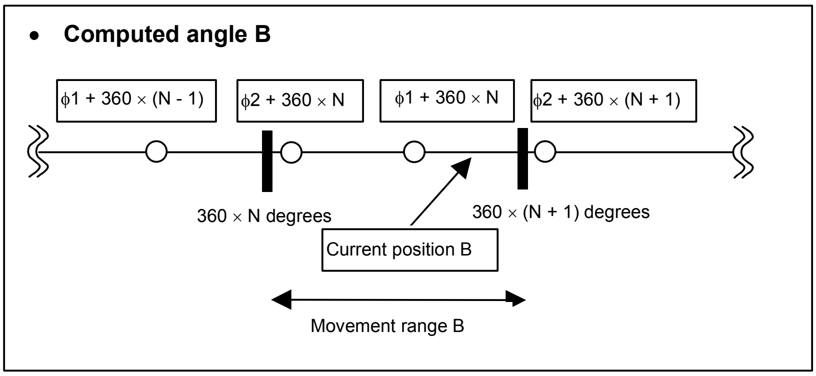
When parameter PRI (No.19608#5) is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

> ⚠ **CAUTION**
> 1 If the lower limit of the movement range is larger than the upper limit, alarm PS5459 occurs when G43.5 is specified.
> 2 If no "computed angle" is found within the movement range because the range is too small, alarm PS5459 occurs.
> 3 If 0 is set for both parameters that specify the upper and lower limits of the movement range, the tool operates assuming that there is no range specification.
> 4 When the rotary axis rollover function or rotary axis control function is used (in which case, set parameter No.1260 (amount of movement per rotation of the rotary axis) to 360 degrees), the tool does not move beyond 0 degree (360 degrees) (does not take the shortcut) if the movement range is set between 0 and 360 degrees. Also, do not specify a negative value or a value larger than 360 degrees for the movement range.

An example of the "movement judgement" process is given below.
Assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A $\theta 1$ degree; B $\phi 1$ degree)
(A $\theta 2$ degrees; B $\phi 2$ degrees) where $\theta 1 \leq \theta 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
Assume that the current positions and movement ranges of rotary axis A (master) and rotary axis B (slave) are as shown in the following figure.



"Computed angle of rotary axis A and its current position and movement range"



"Computed angle of rotary axis B and its current position and movement range"

When the two axes have a positional relationship as shown in the figure, the output angle of rotary axis A is ($\theta 2 + 360 \times N$) degrees and that of rotary axis B is ($\phi 2 + 360 \times N$) degrees (when parameter PRI (No.19608#5) is set to 0).
More concretely, from the computed angles obtained for rotary axis A, the nearest angle within the movement range, i.e. $\theta 2 + 360 \times N$ degrees, is first adopted. Then, from the computed angles obtained for rotary axis B, the angle belonging to the same group as θ2, i.e. $\phi 2 + 360 \times N$, is adopted.

Note that, in this example, the output angles and moving direction differ depending on whether the movement range is specified or not (0 to 360 degrees), even if N is set to 0 and coordinates are rounded to 0 to 360 degrees.
Namely, if the movement range is not specified, $\theta 1 + 360$ degrees nearest to the current position is adopted as the computed angle for rotary axis A and, from the computed angles belonging to the same group as θ1, ϕ1 degrees nearest to the current position is adopted as the computed angle for rotary axis B. Rotary axis A moves in the plus direction. As its coordinate is rounded to 360 degrees, rotary axis A reaches θ1 degrees while moving in the plus direction.

By contrast, when the movement range is set to 0 to 360 degrees, the output angles are (A θ2 degrees; B ϕ2 degrees). Neither rotary axis A nor B moves in a way that it exceeds 0 degree (360 degrees).

Operation examples

- In the case of a tool rotation type machine

Explanations are given below assuming a machine configuration in which a tool rotation axis that turns around the Y-axis is located beneath another tool rotation axis that turns around the Z-axis. (See Fig. 21.1 (j).)

If linear interpolation is specified for the X-, Y-, and Z-axes when a workpiece coordinate system is used as the programming coordinate system, control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the tool rotates.
Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

In the case of a machine having two tool rotation axes, the table does not rotate with respect to the workpiece coordinate system even if the rotary axes move. Therefore, the programming coordinate system always matches the workpiece coordinate system, regardless of whether parameter WKP (No.19696#5) is set to 0 or 1.

Examples)
For type 1:

```
O100 (Sample Program1) ;
N1 G00 G90 B0 C0 ;
N2 G55 ;                          Prepares the programming coordinate
                                  system.
N3 G43.4 H01 ;                    Starts tool center point control.
                                  H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;      Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 C60.0 B45.0 F500 ; Linear interpolation
N6 G49;                           Cancels tool center point control.
N7 M30;
```

For type 2:

```
O100 (Sample Program1) ;
N1 G00 G90 B0 C0 ;
N2 G55 ;                  Prepares the programming coordinate system.
N3 G43.4 H01 ;            Starts tool center point control.
                          H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ; Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 I1.0 J1.732 K2.0 F500 ; Linear interpolation
N6 G49;                   Cancels tool center point control.
N7 M30;
```

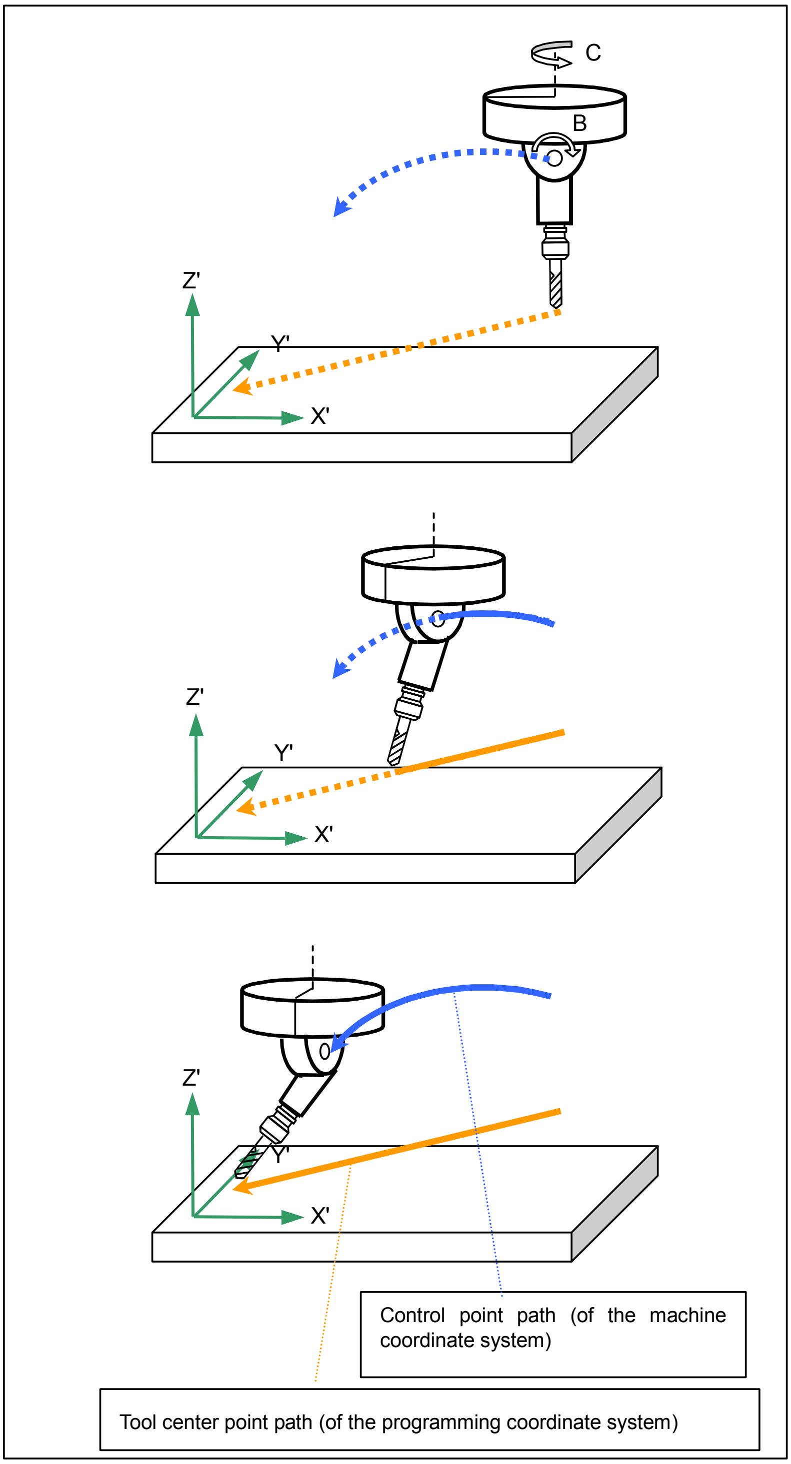


Fig. 21.1 (j) Example for a tool rotation type machine

- In the case of a table rotation type machine

Explanations are given below assuming a machine configuration (trunnion) in which a rotation table that turns around the Y-axis is located above another table rotation axis that turns around the X-axis. (See Fig. 21.1 (k).)

If linear interpolation is specified for the X-, Y-, and Z-axes on the programming coordinate system and if the rotary axis that moves the rotation table is specified (in the case of type 1) or the tool direction is specified (in the case of type 2), control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the rotation table rotates.
Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

Example)
When type 1 is selected and the coordinate system fixed on the table is used as the programming coordinate system
(Parameter WKP (No.19696#5) = 0):

```
O200 (Sample Program2) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                        Prepares the programming coordinate
                                system.
N3 G43.4 H01 ;                  Starts tool center point control.
                                H01 is the tool compensation number.
N4 G00 X20.0 Y100.0 Z0 ;        Moves to the start point.
N5 G01 X10.0 Y20.0 Z30.0 A60.0 B45.0 F500 ; Linear interpolation
N6 G49;                         Cancels tool center point control.
N7 M30;
```

When type 1 is selected and the workpiece coordinate system is used as the programming coordinate system
([Parameter WKP (No.19696#5) = 1):

```
O200 (Sample Program2) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                        Prepares the programming coordinate
                                system.
N3 G43.4 H01 ;                  Starts tool center point control.
                                H01 is the tool compensation number.
N4 G00 X20.0 Y100.0 Z0 ;        Moves to the start point.
N5 G01 X7.574 Y47.247 Z83.052 A60.0 B45.0 F500 ;
                                Linear interpolation
N6 G49;                         Cancels tool center point control.
N7 M30;
```

For type 2 (when the coordinate system fixed on the table is used as the programming coordinate system (only when parameter WKP (No.19696#5) is set to 0)):

```
O200 (Sample Program2) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                        Prepares the programming coordinate
                                system.
N3 G43.5 H01 ;                  Starts tool center point control.
                                H01 is the tool compensation number.
N4 G00 X20.0 Y100.0 Z0 ;        Moves to the start point.
N5 G01 X10.0 Y20.0 Z30.0 I-1.0 J2.449 K1.0 F500 ;
                                Linear interpolation
N6 G49;                         Cancels tool center point control.
N7 M30;
```

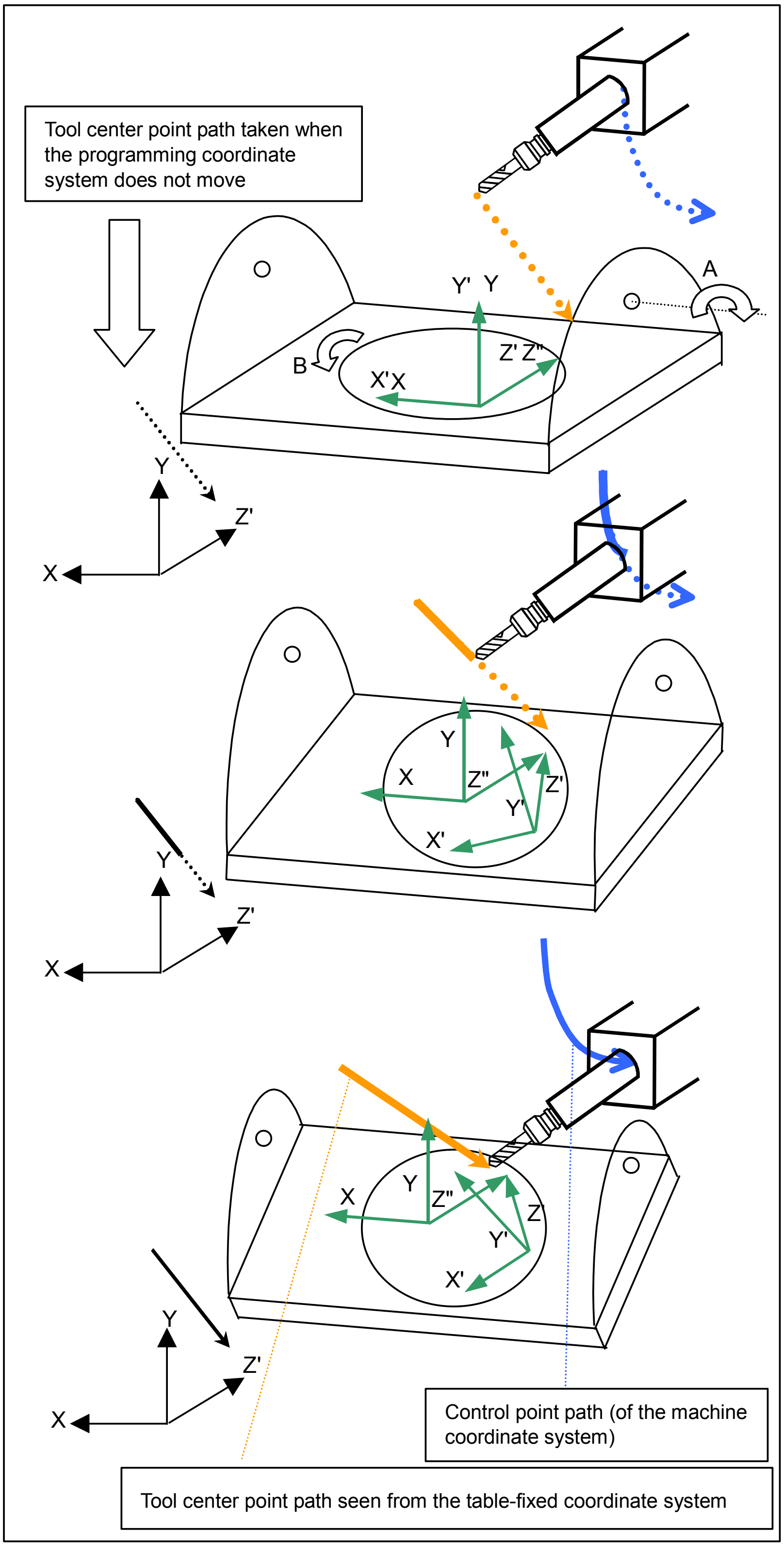


Fig. 21.1 (k) Example for a table rotation type machine

- In the case of a mixed type machine

Explanations are given below assuming a mixed type machine configuration that has one table rotation axis (which turns around the X-axis) and one tool rotation axis (which turns around the Y-axis). (See Fig. 21.1 (l).)

If linear interpolation is specified for the X-, Y-, and Z-axes on the programming coordinate system and if the rotary axis that moves the rotation table and the tool rotation axis are specified (in the case of type 1) or the tool direction is specified (in the case of type 2), control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the rotation table and tool rotate.
Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

Example)
When type 1 is selected and the coordinate system fixed on the table is used as the programming coordinate system
(Parameter WKP (No.19696#5) = 0):

```
O300 (Sample Program3) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                        Prepares the programming coordinate
                                system.
N3 G43.4 H01 ;                  Starts tool center point control.
                                H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;    Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 A60.0 B45.0 F500;
                                Linear interpolation
N6 G49 ;                        Cancels tool center point control.
N7 M30;
```

When type 1 is selected and the workpiece coordinate system is used as the programming coordinate system
(Parameter WKP (No.19696#5) = 1):

```
O300 (Sample Program3) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                        Prepares the programming coordinate
                                system.
N3 G43.4 H01 ;                  Starts tool center point control.
                                H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ; Moves to the start point.
N5 G01 X5.0 Y48.170 Z-79.772 A60.0 B45.0 F500;
                                Linear interpolation
N6 G49 ;                        Cancels tool center point control.
N7 M30;
```

For type 2 (when the coordinate system fixed on the table is used as the programming coordinate system (only when parameter WKP (No.19696#5) is set to 0)):

```
O300 (Sample Program3) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                        Prepares the programming coordinate
                                system.
N3 G43.5 H01 ;                  Starts tool center point control.
                                H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ; Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 I2.0 J1.732 K1.0 F500;
                                Linear interpolation
N6 G49 ;                        Cancels tool center point control.
N7 M30;
```

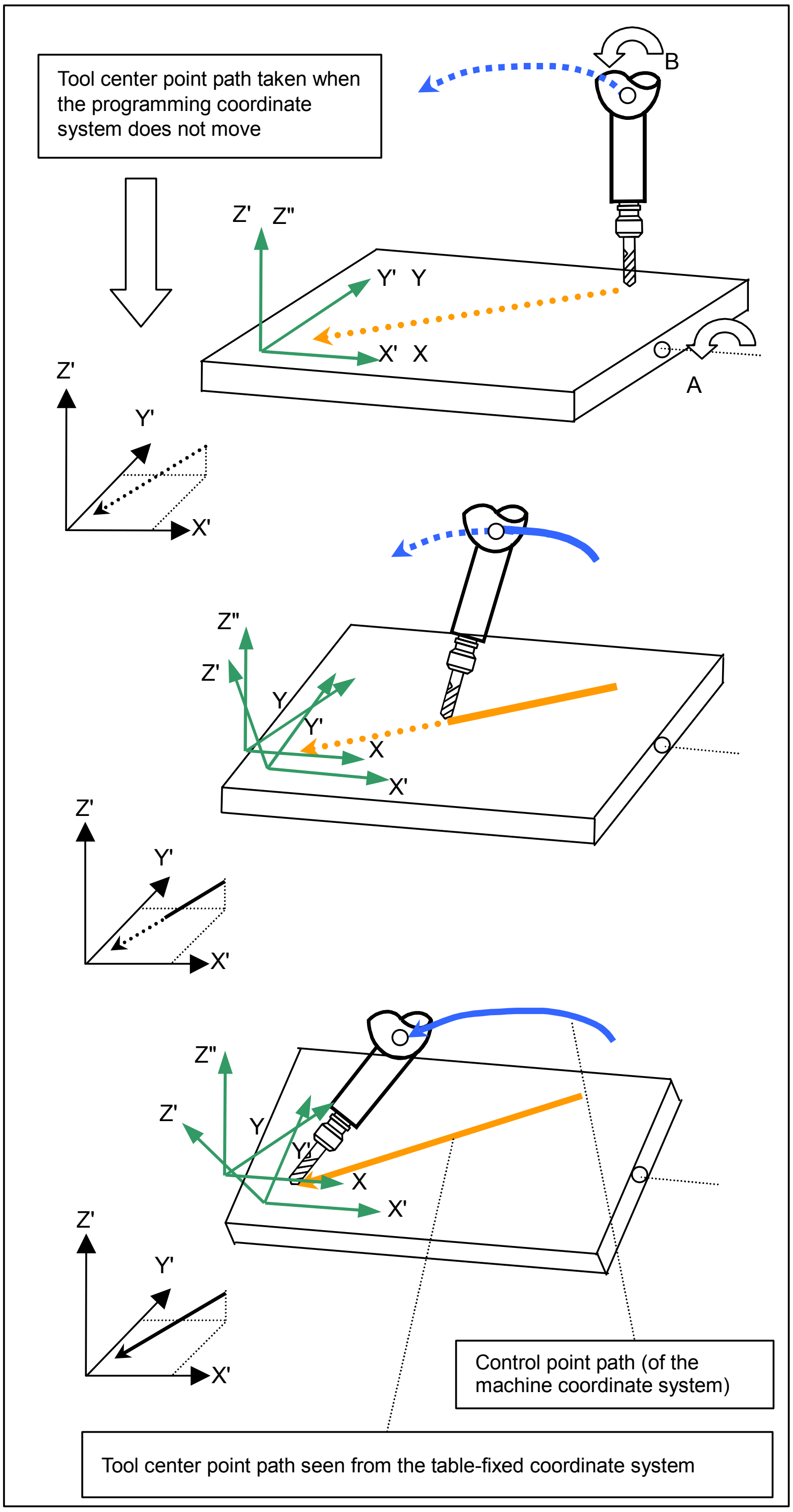


Fig. 21.1 (l) Example for a mixed type machine

- When linear interpolation is performed during tool center point control

Examples are given below in which each 100-mm-long side of an equilateral triangle is cut at B-axis angles of 0, 30 to 60, and 60 degrees, respectively.

Example)
When type 1 is selected and the table-fixed coordinate system is used as the programming coordinate system:

O400 (Sample Program4) ;	
N10 G55 ;	Prepares the programming coordinate system.
N20 G90 X50.0 Y-70.0 Z300.0 B0 C0 ;	Moves to the initial position.
N30 G01 G43.4 H01 Z20.0 F500. ;	Starts tool center point control. Moves to the approaching position. H01 is the tool compensation number.
N40 X28.868 Y-50.0 Z10.0 B30.0 ;	The Z-axis height on the machining plane is 10.0.
N50 Y50.0 ;	
N60 B45.0 C120.0 ;	
N70 X-57.735 Y0 B60.0 C180.0 ;	Moves X and Y while operating both B- and C-axes.
N80 C240.0 ;	
N90 X28.868 Y-50.0 ;	
N100 X50.0 Y-70.0 Z20.0 B0 C360.0 ;	X, Y, and Z are approaching positions. The rotary axes remain at their original positions.
N110 G49 Z300.0 ;	Cancels tool center point control. Moves the Z-axis to its initial position.
N120 M30;	

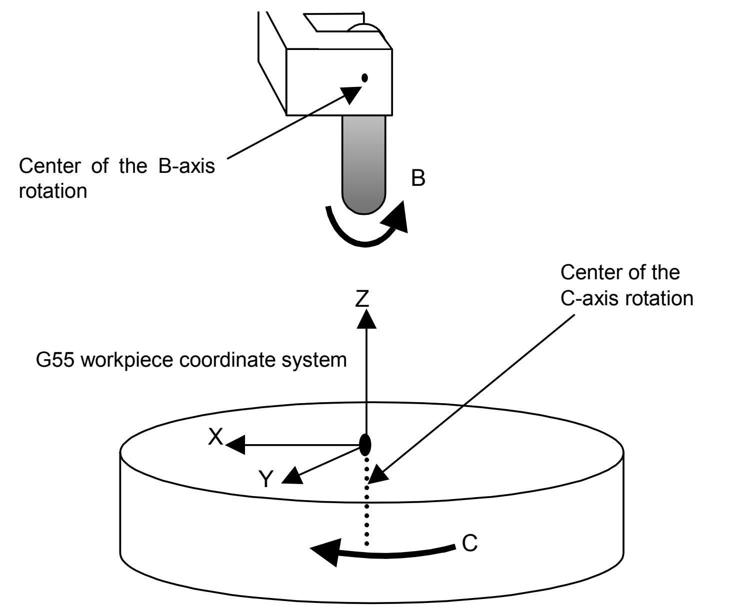
When type 1 is selected and the workpiece coordinate system is used as the programming coordinate system
(Note that the values of N60 to N90 are different from those specified in the preceding example.):

O400 (Sample Program4) ;	
N10 G55 ;	Prepares the programming coordinate system.
N20 G90 X50.0 Y-70.0 Z300.0 B0 C0 ;	Moves to the initial position.
N30 G01 G43.4 H01 Z20.0 F500. ;	Starts tool center point control. Moves to the approaching position. H01 is the tool compensation number.
N40 X28.868 Y-50.0 Z10.0 B30.0 ;	The Z-axis height on the machining plane is 10.0.
N50 Y50.0 ;	
N60 Y-50.0 B45.0 C120.0 ;	
N70 X57.735 Y0 B60.0 C180.0 ;	Moves X and Y while operating both B- and C-axes.
N80 X28.868 Y-50.0 C240.0 ;	
N90 Y50.0 ;	
N100 X50.0 Y-70.0 Z20.0 B0 C360.0 ;	X, Y, and Z are approaching positions. The rotary axes remain at their original positions.
N110 G49 Z300.0 ;	Cancels tool center point control. Moves the Z-axis to its initial position.
N120 M30;	

When type 2 is selected and the table-fixed coordinate system is used as the programming coordinate system:

O400 (Sample Program4) ;	
N10 G55 ;	Prepares the programming coordinate system.
N20 G90 X50.0 Y-70.0 Z300.0 B0 C0 ;	Moves to the initial position.
N30 G01 G43.5 H01 Z20.0 F500. ;	Starts tool center point control. Moves to the approaching position. H01 is the tool compensation number.
N40 X28.868 Y-50.0 Z10.0 I1.0 K1.732 ;	The Z-axis height on the machining plane is 10.0.
N50 Y50.0 ;	
N60 I-0.5 J0.866 K1.0 ;	
N70 X-57.735 Y0 I-1.732 K1.0 ;	Moves X and Y while operating both B- and C-axes.
N80 I-0.866 J1.5 K1.0 ;	
N90 X28.868 Y-50.0 ;	
N100 X50.0 Y-70.0 Z20.0 K1.0 ;	X, Y, and Z are approaching positions. The rotary axes remain at their original positions.
N110 G49 Z300.0 ;	Cancels tool center point control. Moves the Z-axis to its initial position.
N120 M30;	

- **Mixed type machine (tool rotation axis = B-axis; table rotation axis = C-axis; tool axis = Z direction)**



Machine configuration for the example

The following figure illustrates the position of the workpiece, as well as the position of the tool head (relative to the workpiece), as seen from the table-fixed programming coordinate system in the +Z direction.

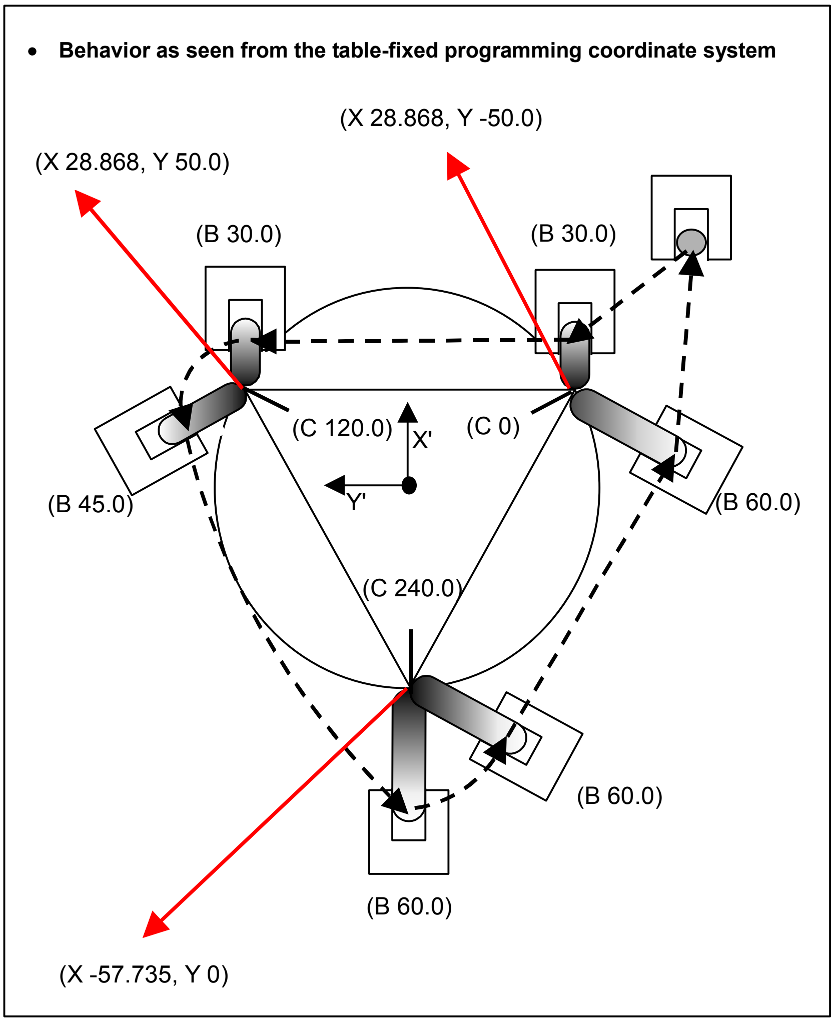
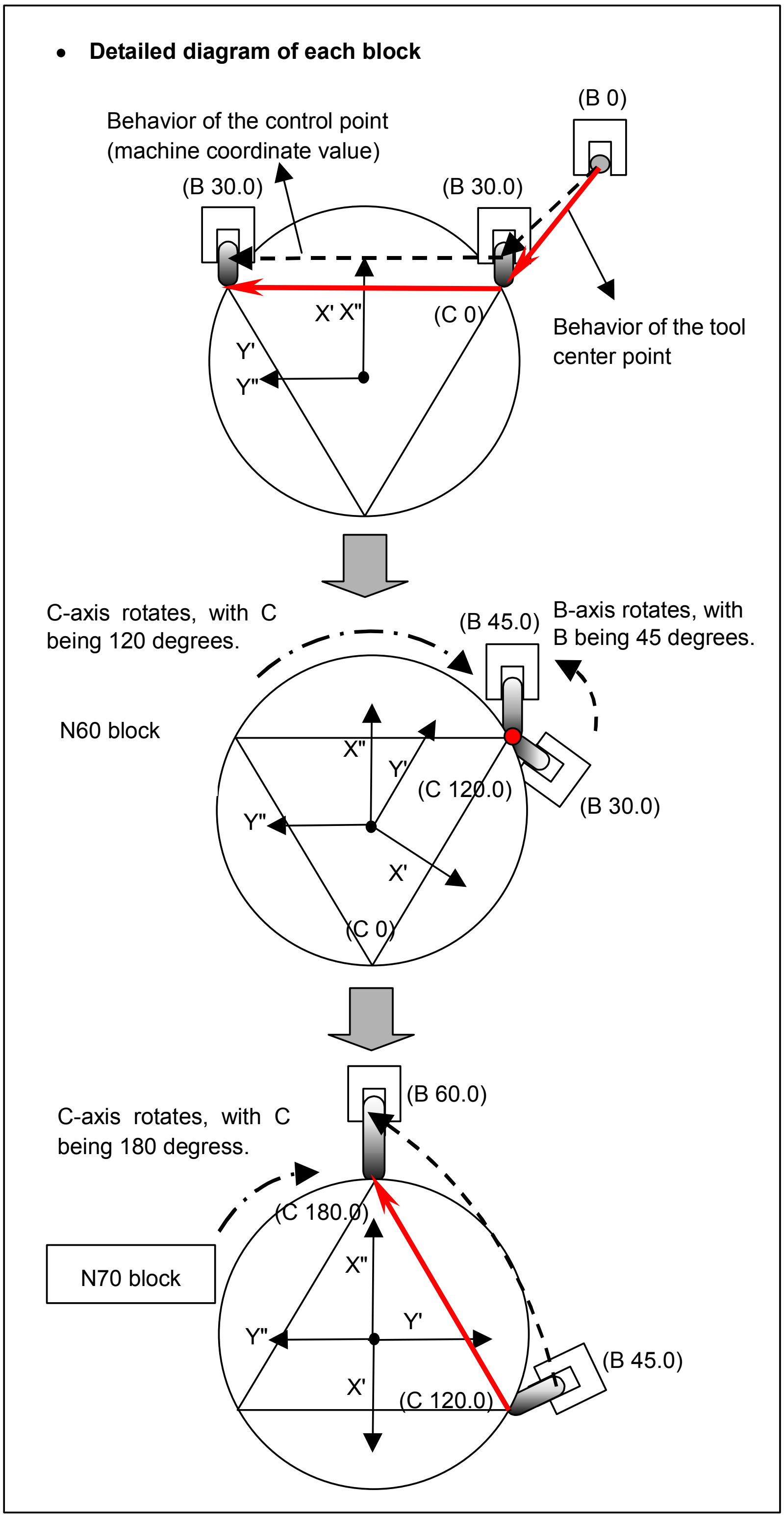
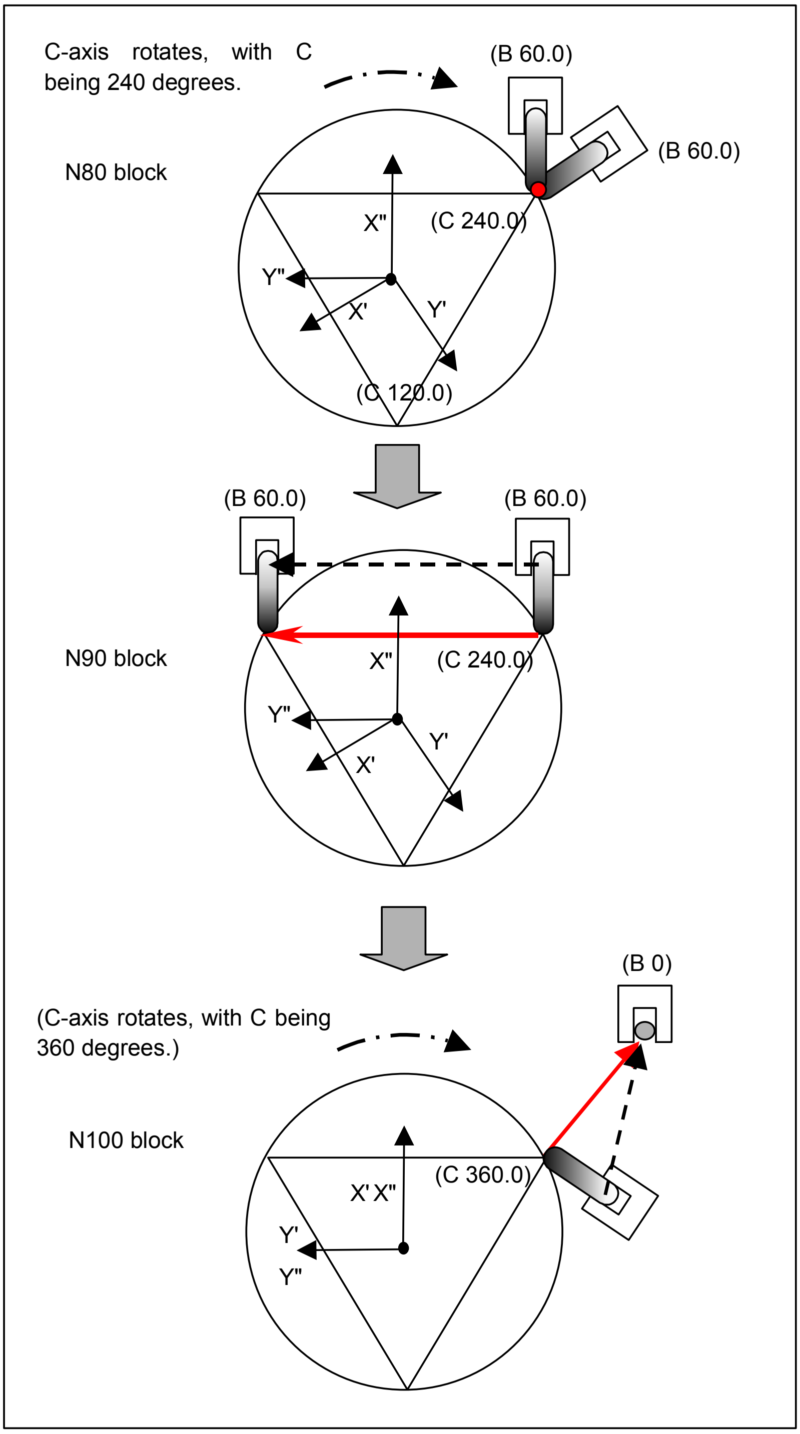


Illustration of the example (1)



Detailed diagram of each block (1)



Detailed diagram of each block (2)

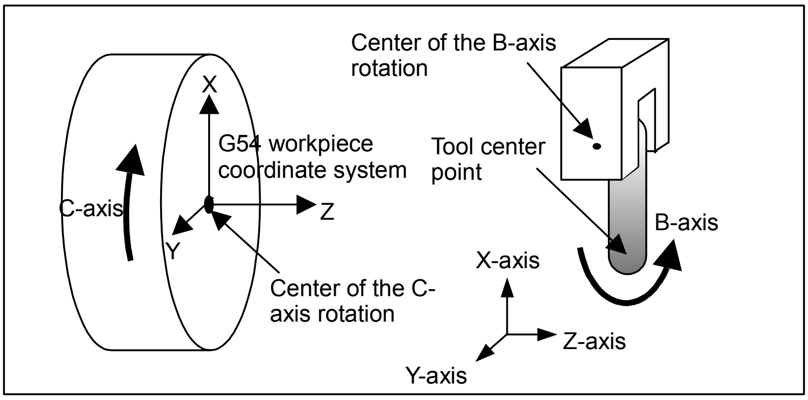
- When circular interpolation is performed during tool center point control

In this example, one of the three sides of an equilateral triangle, each being 100 mm long side, is specified as a straight line and the other two are specified as arcs, and each side is cut at B-axis angles of -60, -45 to -30, and -30 degrees, respectively. (For the X-axis, its radius is specified.)

Example)

The program sample given below assumes that the table-fixed coordinate system is used as the programming coordinate system. When the workpiece coordinate system is used as the programming coordinate system (when parameter No.19696#5 (WKP) is set to 1), the same shape can be obtained by specifying the values shown in parentheses.

```
N001 T0000 ;                    Cancels the tool offset.
N002 G54 ;                      Selects the workpiece coordinate system.
N003 G90 X50.0 Y-70.0 Z300.0 B-90.0 C0.0 ;   Moves to the initial position.
                                The tool tip faces the Z- direction at B-axis
                                angle of -90 degrees.
N020 G01 G43.4 H01              Starts tool center point control.
N021 Z20.0 ;                    Moves to the approaching position.
N022 X28.868 Y-50.0 Z10.0 B-60.0 ;  The Z-axis height on the machining
                                    plane is 10.0.
N031 Y50.0 ;
N032 B-45.0 C90.0 ;
(N032 X50.0 Y-28.868 B-45.0 C90.0 ;)
N033 G17 G03 X-57.735 Y0.0 J-100.0 B-30.0 C150.0 ;
                                Moves X and Y while operating both B-
                                and C-axes.
(N033 G17 G03 X50.0 Y28.868 I-100.0 B-30.0 C150.0 ;)
N034 G01 B-30.0 C210.0 ;
(N034 G01 X50.0 Y-28.867 B-30.0 C210.0 ;)
N035 G03 X28.868 Y-50.0 I86.603 J50.0 C270.0 ;
(N035 G03 X50.0 Y28.868 I-100.0 C270.0 ;)
N041 G01 X50.0 Y-70.0 Z20.0 B-90.0 C360.0 ;
                                X, Y, and Z are approaching positions.
                                The rotary axes remain at their original
                                positions.
N050 G49                        Cancels tool center point control.
N051 Z300.0 ;                   Moves the Z-axis to its initial position.
```



Machine configuration for the circular interpolation example

The following figure illustrates the relative positional relationship between the workpiece and the tool head and their positions, as seen from the +Z direction. The X and Y coordinate values shown in the figure are those on the table-fixed programming coordinate system (which rotates as the C-axis turns).

- **Behavior as seen from the table-fixed programming coordinate system**

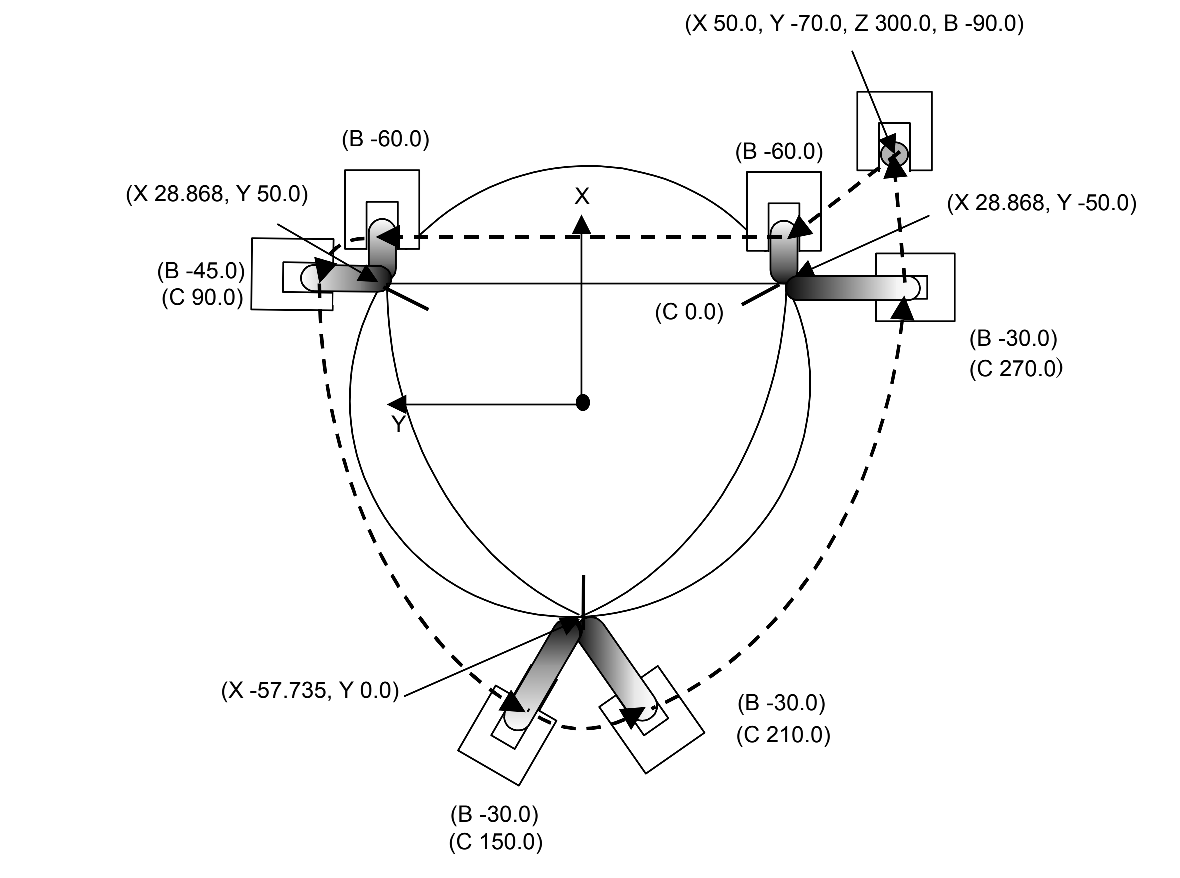
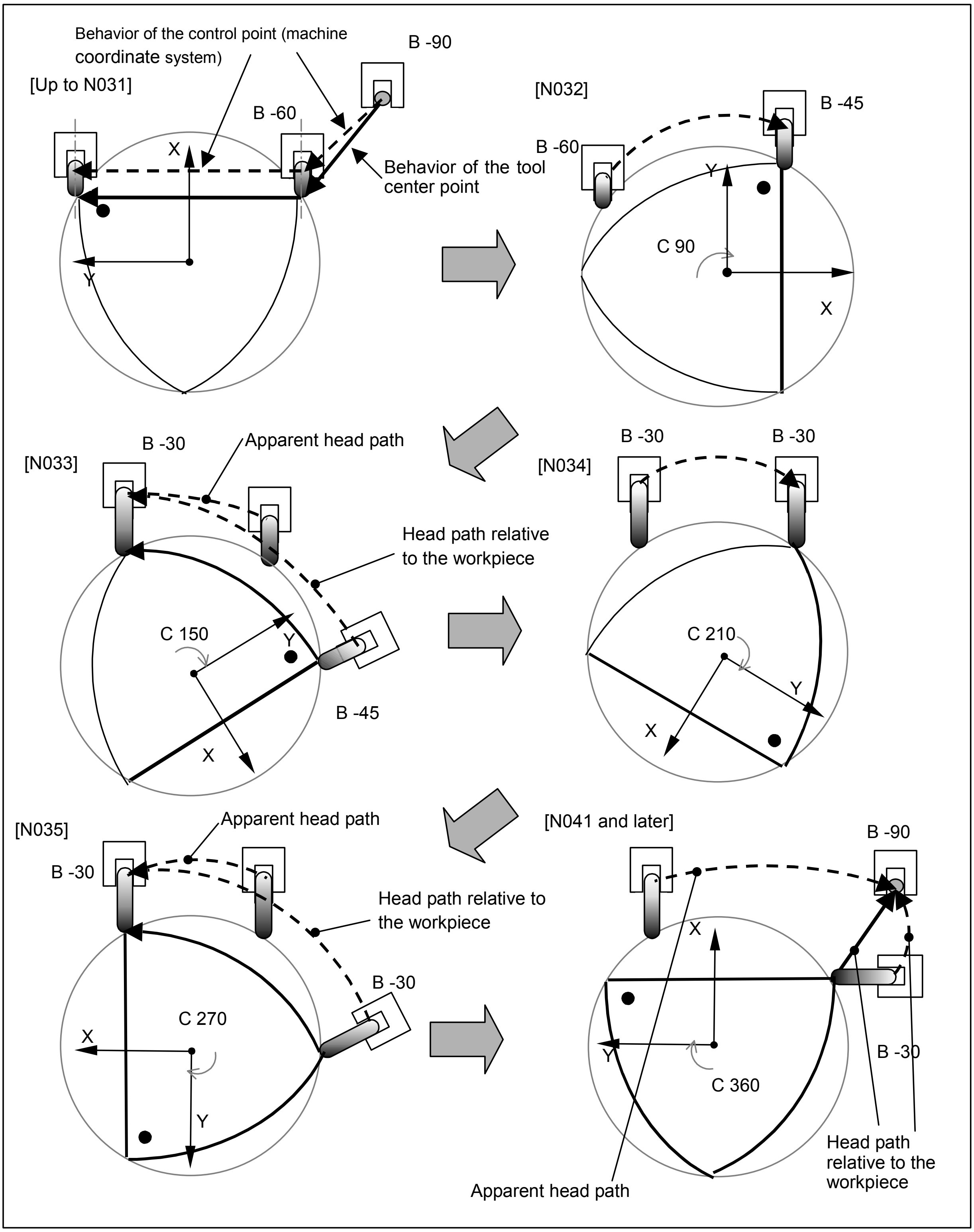


Illustration of the example



Detailed diagram of each block

Limitation

- Manual intervention

In tool center point control mode, do not perform manual intervention. Otherwise, an alarm is generated.

- Hypothetical axis of a table rotation axis

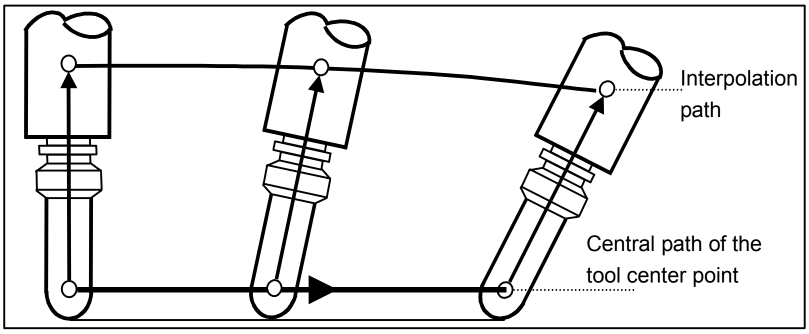
When a table rotation axis is set as a hypothetical axis, tool center point control is performed on the assumption that the table rotation axis is at 0 degrees.

- Deceleration at a corner

During tool center point control, the control point may move on a curved line even if a straight line is specified. Some commands may cause the tool center point to make a sharp turn.
For this reason, the tool may be decelerated if a small value is set as the permissible speed difference (parameter No. 1783) or the permissible acceleration (parameter No. 1660 or 1737) for a corner.

- Look-ahead pre-interpolation acceleration/deceleration

When using tool center point control, use look-ahead acceleration/deceleration before interpolation at the same time. If look-ahead acceleration/deceleration before interpolation is not used, alarm PS5420 is issued.
For details, see "NOTE" in "Commands that can be specified during tool center point control" in "Explanation."



In the above example, the feedrate is controlled so that the tool center point moves at a specified feedrate, so the feedrate on the interpolation path is equal to or higher than the specified feedrate. At this time, look-ahead acceleration/deceleration before interpolation allows the feedrate to be clamped so that the feedrate on the interpolation path does not exceed the maximum cutting feedrate or maximum rapid traverse rate.

- Cutter compensation for 5-axis machining

When tool center point control is exercised together with cutter compensation for 5-axis machining on a machine of mixed type or table rotation type, specify a value in the workpiece coordinate system by setting the parameter WKP (No. 19696) to 1.
In that case, when specifying cutter compensation for 5-axis machining before tool center point control, specify the cancellation of cutter compensation for 5-axis machining after canceling tool center point control (example 1). When specifying tool center point control before cutter compensation for 5-axis machining, specify the cancellation of tool center point control after canceling three-dimensional cutter compensation (example 2).

```
Example 1

G41.2   D1
  :
G43.4   H1
  :
G49
  :
G40
```

```
Example 2

G43.4   H1
  :
G41.2   D1
  :
G40
  :
G49
```

When specifying cutter compensation for 5-axis machining first, the block for canceling tool center point control controls buffering. Note that, in a block preceding the G49 block, the compensation vector for cutter compensation for 5-axis machining is directed toward the vertical direction of movement.

- Parallel axis control

When exerting tool center point control together with parallel axis control, make sure that the master and slave axes are properly aligned and keep the parking signal off.

- Programmable mirror image

Note the following points when making a programmable mirror image:

- In the case of tool center point control of type 1
  Mirroring the linear axis alone does not create a mirror image for the rotary axis. To make the direction of the tool symmetrical, it is necessary to make a mirror image for the rotary axis as well.
- In the case of tool center point control of type 2
  When parameter MIR (No.19608#6) is set to 0, mirroring the linear axis alone does not create a mirror image for I, J, K. Note that the inclination direction specified by Q is the movement direction of the tool center point after mirroring.
  When parameter MIR (No.19608#6) is set to 1, mirroring the linear axis automatically creates a mirror image for I, J, K.
  A mirror image cannot be put on the rotary axis directly.

- Specifiable G codes

The G codes that can be specified in the tool center point control mode are listed below.
Specifying a G code other than these codes results in alarm PS5421.

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation / helical interpolation (G02/G03)
- Dwell (G04)
- Exact stop (G09)
- Programmable data input (G10)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Stored stroke check function (G22/G23)
- Cutter or tool nose radius compensation : preserve vector (G38)
- Cutter or tool nose radius compensation : corner circular interpolation (G39)
- Cutter compensation : cancel (G40)
- Cutter or tool nose radius compensation /
  Three-dimensional cutter compensation (G41/G42)
- Cutter compensation for 5-axis machining
  (G41.2/G42.2/G41.4/G42.4/G41.5/G42.5)
- Tool length compensation cancel (G49)
- Scaling (G50/G51)
- Programmable mirror image (G50.1/G51.1)
- Exact stop mode (G61)
- Automatic corner override mode (G62)
- Tapping mode (G63)
- Cutting mode(G64)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Figure copy (G72.1/G72.2)
- Absolute programming (G90)
- Incremental programming (G91)

M

- Tool offset increase (G45)
- Tool offset decrease (G46)
- Tool offset double increase (G47)
- Tool offset double decrease (G48)

- Modal G codes that allow specification of tool center point control

Tool center point control can be specified in the modal G code states listed below.
In a modal state other than the following modal G codes, specifying tool center point control results in alarm PS5421:

- Modal G codes included in "Specifiable G codes" described previously
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Polygon turning cancel (G50.2)
- Workpiece coordinate system 1 selection (G54 to G59)
- Canned cycle cancel (G80)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Coordinate system rotation start or 3-dimensional coordinate conversion mode on (G69)
- Feed per minute (G94)
- Polar coordinate interpolation mode cancel (G113)

T

- Mirror image for double turret off/balanced cutting mode cancel (G69)
- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69.1)
- Feed per minute (G98 (G94))

- Specification of axes not relating to tool center point control

Axes not relating to tool center point control cannot be specified. If such an axis is specified, alarm PS5421 is issued.

- Linear axes under tool center point control

The basic three axes set in parameter No. 1022 are regarded as the three linear axes for tool center point control. Axes parallel to the basic three axes cannot be used as the linear axes for tool center point control.
If the basic three axes are not set in parameter No. 1022, alarm PS5459 is issued.

21.2 TILTED WORKING PLANE COMMAND

Overview

Programming for creating holes, pockets, and other figures in a datum plane tilted with respect to the workpiece would be easy if commands can be specified in a coordinate system fixed to this plane (called a feature coordinate system). This function enables commands to be specified in the feature coordinate system. The feature coordinate system is defined in the workpiece coordinate system.
For explanations about the relationship between the feature coordinate system and workpiece coordinate system, see Fig. 21.2 (a).

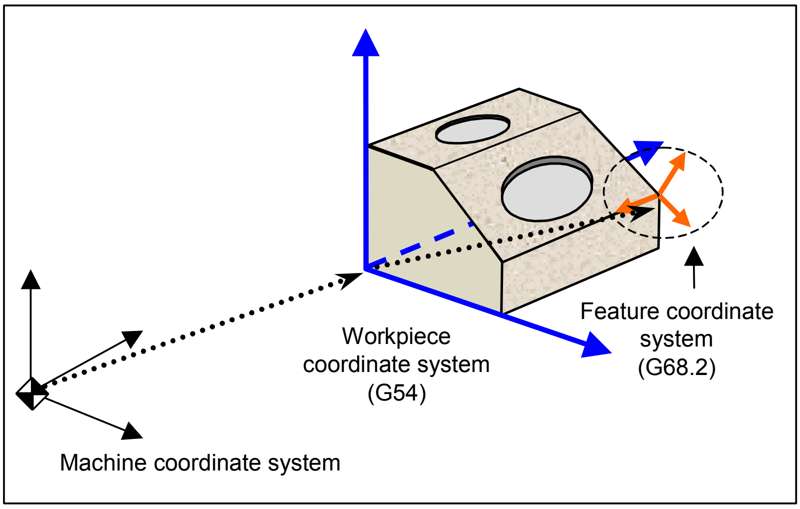


Fig. 21.2 (a) Feature coordinate system

The G68.2 command causes the programming coordinate system to switch to the feature coordinate system. The commands in all subsequent blocks are assumed to be specified in the feature coordinate system until G69 appears.
If G68.2 specifies the relationship between the feature coordinate system and the workpiece coordinate system, G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction even if no angle is specified for the rotary axis. (See Fig. 21.2 (c).)
For explanations about the tool axis direction, see Fig. 21.2 (b).

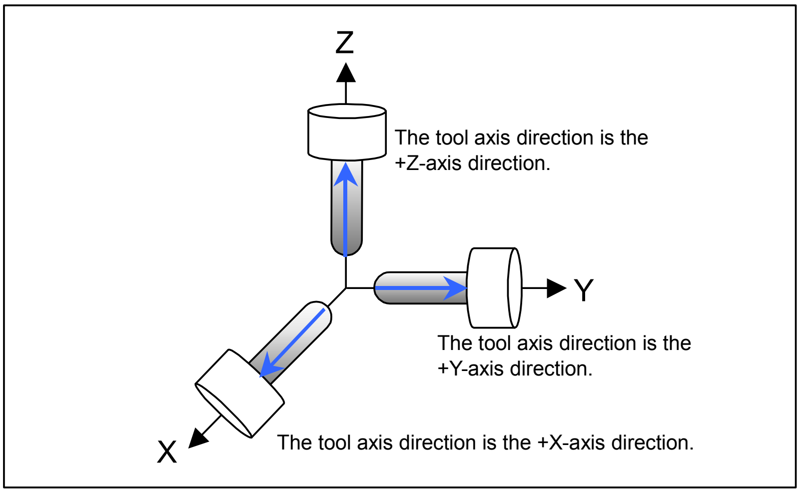
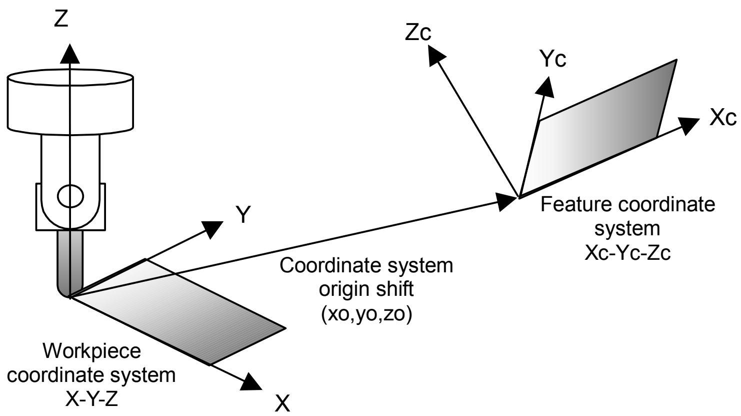


Fig. 21.2 (b) Tool axis direction

This function regards the direction normal to the machining plane as the +Z-axis direction of the feature coordinate system. After the G53.1 command, the tool is controlled so that it remains perpendicular to the machining plane.

- **Only G68.2 is specified**



- **G53.1 is specified after G68.2**

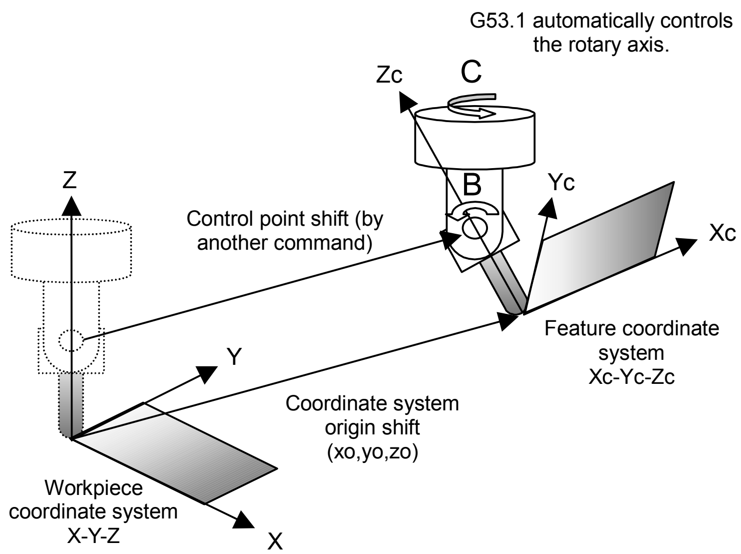


Fig. 21.2 (c) G68.2 and G53.1 commands

This function is applicable to the following machine configurations. (See Fig. 21.2 (d).)

<1> Tool rotation type machine controlled with two tool rotation axes
<2> Table rotation type machine controlled with two table rotation axes
<3> Mixed-type machine controlled with one tool rotation axis and one rotary axis

The function can also be used for a machine configuration in which the rotary axis for controlling the tool does not intersect the rotary axis for controlling the table.

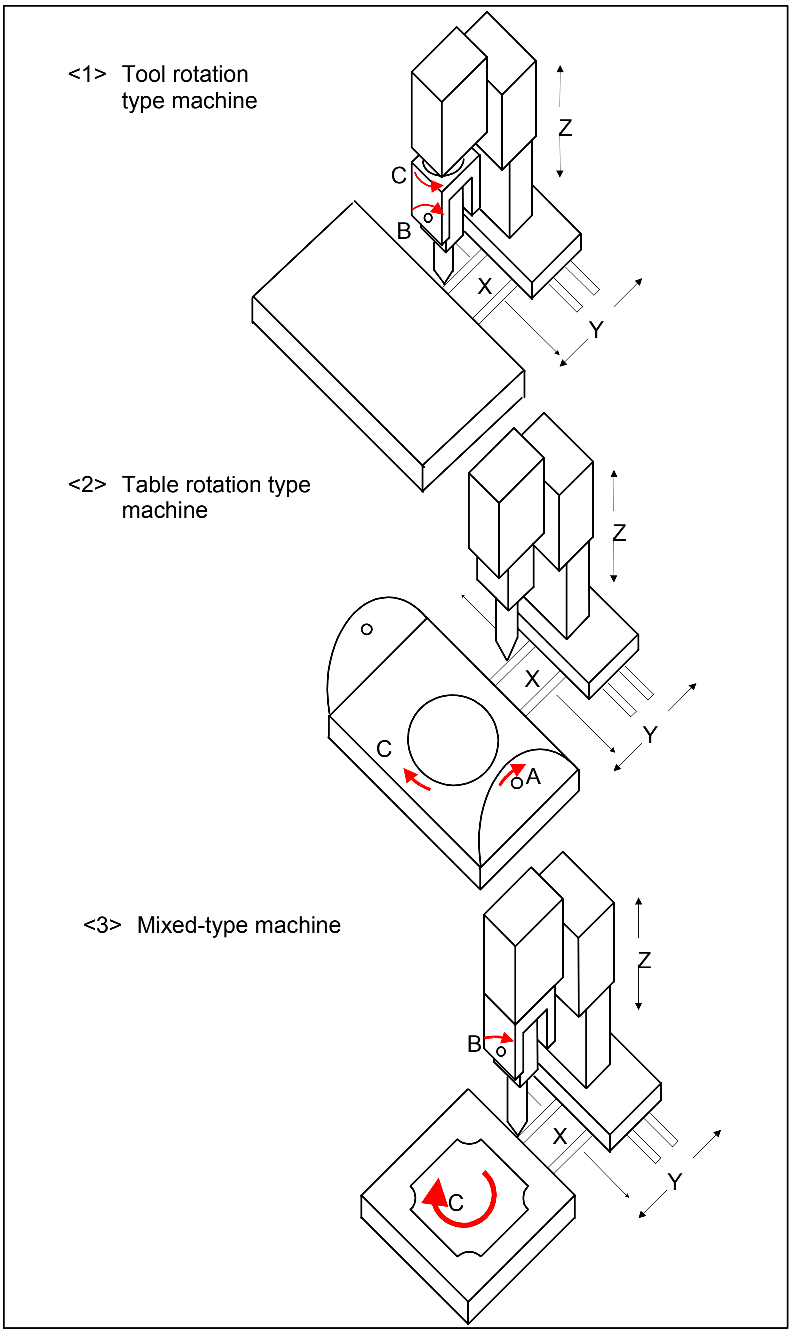


Fig. 21.2 (d) Three types of 5-axis machine

Format

- Feature coordinate system setting (G68.2)

M

G68.2 X $\underline{x0}$ Y $\underline{y0}$ Z $\underline{z0}$ I$\underline{\alpha}$ J$_{\beta}$ K$_{\gamma}$;
Feature coordinate system setting
G69 ; Cancels the feature coordinate system setting.

X, Y, Z : Feature coordinate system origin
I, J, K : Euler's angle for determining the orientation of the feature coordinate system

T

G68.2 X $\underline{x0}$ Y $\underline{y0}$ Z $\underline{z0}$ I$\underline{\alpha}$ J$_{\beta}$ K$_{\gamma}$;
Feature coordinate system setting
G69.1 ; Cancels the feature coordinate system setting.

X, Y, Z : Feature coordinate system origin
I, J, K : Euler's angle for determining the orientation of the feature coordinate system

- Tool axis direction control (G53.1)

G53.1 ; Controls the tool axis direction.

⚠ **CAUTION**

1. G53.1 must be specified in a block after the block that contains G68.2.
   An alarm occurs if G53.1 is specified without G68.2 being specified in a preceding block.
2. G53.1 must be specified in a block in which there is no other command.
3. The rotary axis moves at the maximum rapid traverse federate in the case of rapid traverse and at the specified federate in the case of cutting feed.

Explanation

- Coordinate conversion using an Euler's angle

Coordinate conversion by rotation is assumed to be performed around the workpiece coordinate system origin.
Let the coordinate system obtained by rotating the workpiece coordinate system around the Z-axis by an angle of α degrees be coordinate system 1. Similarly, let the coordinate system obtained by rotating coordinate system 1 around the X-axis by an angle of β be coordinate system 2. The feature coordinate system is the coordinate system obtained by shifting the coordinate system that is obtained by rotating coordinate system 2 around the Z-axis through an angle of γ degrees from the workpiece coordinate system origin by (Xo, Yo, Zo).

Fig. 21.2 (e) shows the relationship between the workpiece coordinate system and the feature coordinate system.
The figure also gives examples of displacement on the X-Y plane.

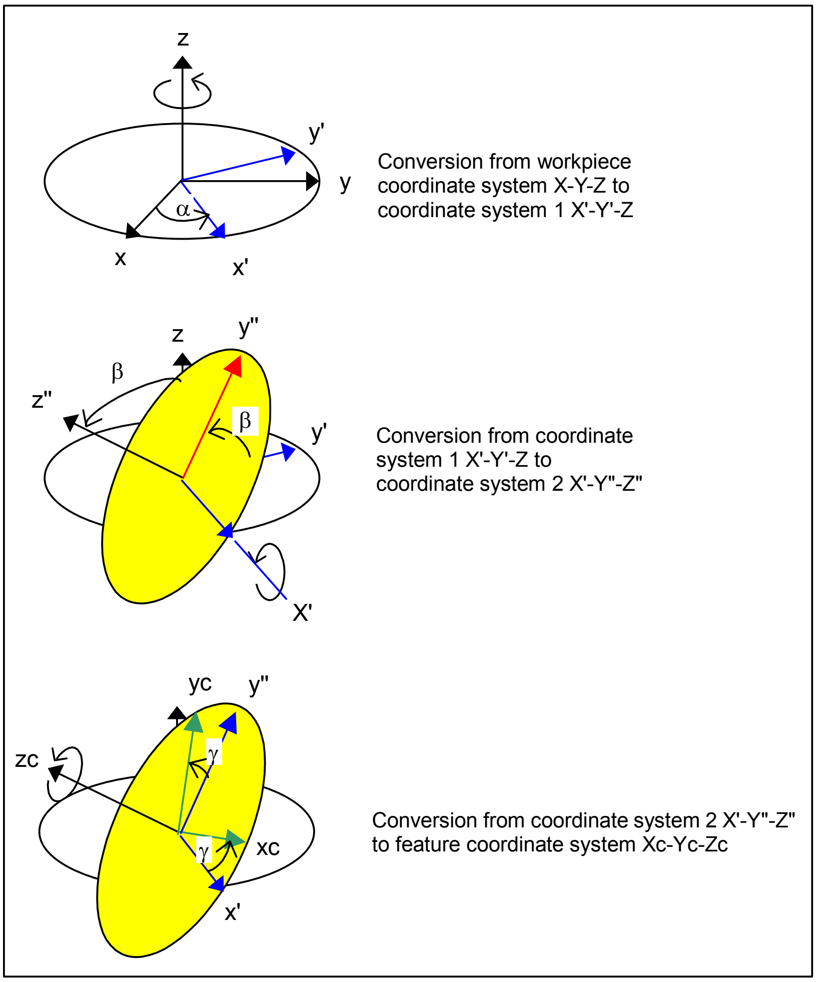


Fig. 21.2 (e) Coordinate conversion using an Euler's angle

- Tool rotation type machine

The following paragraphs describe several cases of the tool rotation type machine operation.

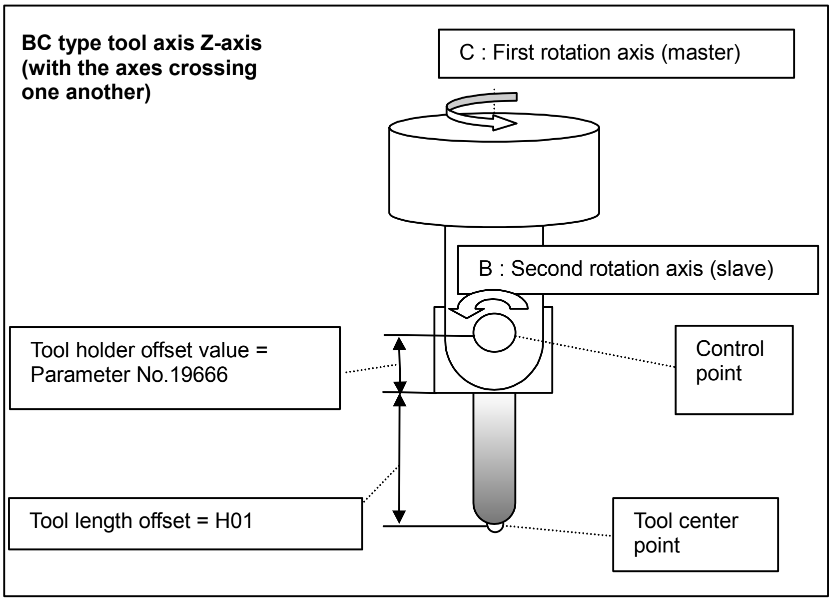
Operation description 1: When G43 (tool length compensation) is specified for a machine with its axes crossing one another

The G53.1 command, when specified after the G68.2 command, automatically controls the rotary axis in such a way that the tool axis will be oriented in the +Z direction of the feature coordinate system.

Example)

```
O100 (Sample Program1) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z30.0 F1000 ;
N5 G53.1 ;
N6 G43 H01 X0 Y0 Z0 ;
N7 …
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
In addition, the tool axis, tool rotation axis B, and tool rotation axis C cross one another.



Block N3 : Defines a feature coordinate system in the workpiece coordinate system.
Block N4 : Shifts the control point to point Z30.0 in the feature coordinate system.
Block N5 : Exerts automatic control over the rotary axes.
Block N6 : Performs tool length compensation in the feature coordinate system.
The tool center point is shifted to the origin of the feature coordinate system.

Fig. 21.2 (f) shows the behavior of the machine when it runs sample program 1.

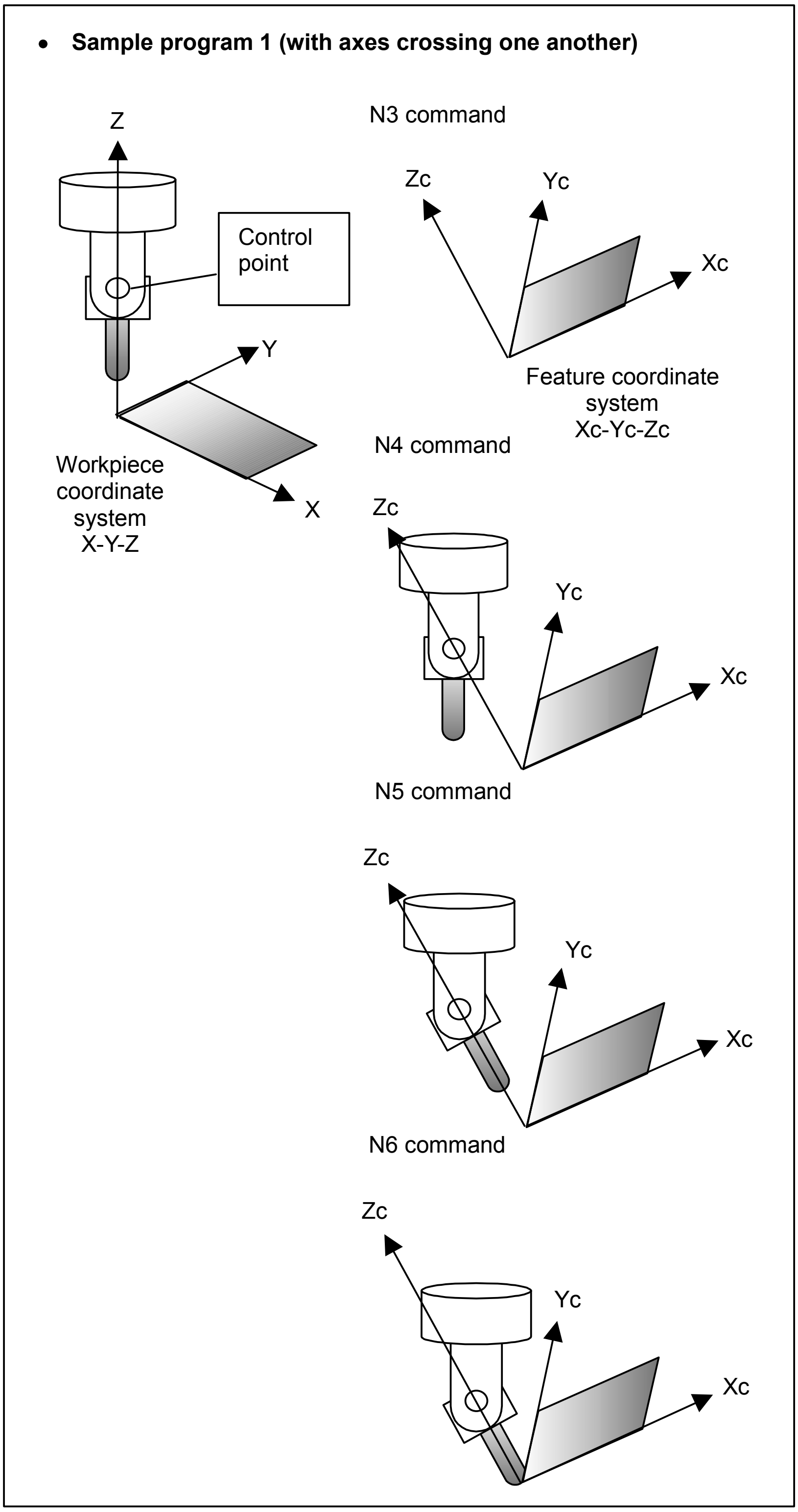
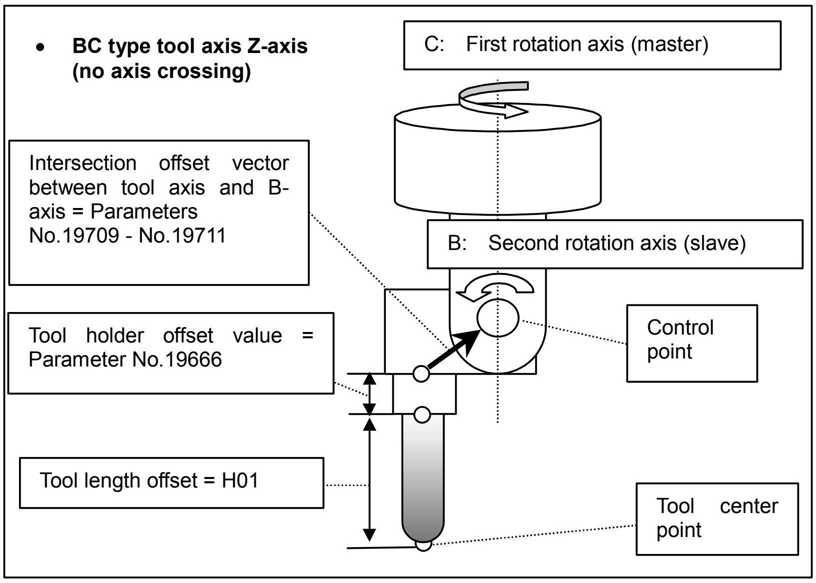


Fig. 21.2 (f) Tool axis direction control 1

Operation description 2: When G43 (tool length compensation) is specified for a machine with no axis crossing

Here is the case where no axis of the machine crosses any other axis.
It is assumed that sample program 1 is used.

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
It is assumed, however, that the tool axis does not cross the B-axis while the B-axis and C-axis cross each other.



Block N4 : Shifts the control point to point Z30.0 in the feature coordinate system.
Block N5 : Exerts automatic control over the rotary axes.
Block N6 : An intersection offset vector between the tool axis and the B-axis with automatic control for rotary axes taken into consideration is output in the feature coordinate system.
Performs tool length compensation in the feature coordinate system.
The tool center point is shifted to the origin of the feature coordinate system.

This is also true when the B-axis does not cross the C-axis.
For explanations about the offset to be applied when the B-axis does not cross the C-axis, see the descriptions about parameters No.19712, No.19713, and No.19714.

Fig. 21.2 (g) shows the behavior of the machine when it runs sample program 1.

- **Sample program 1 (no axis crossing)**

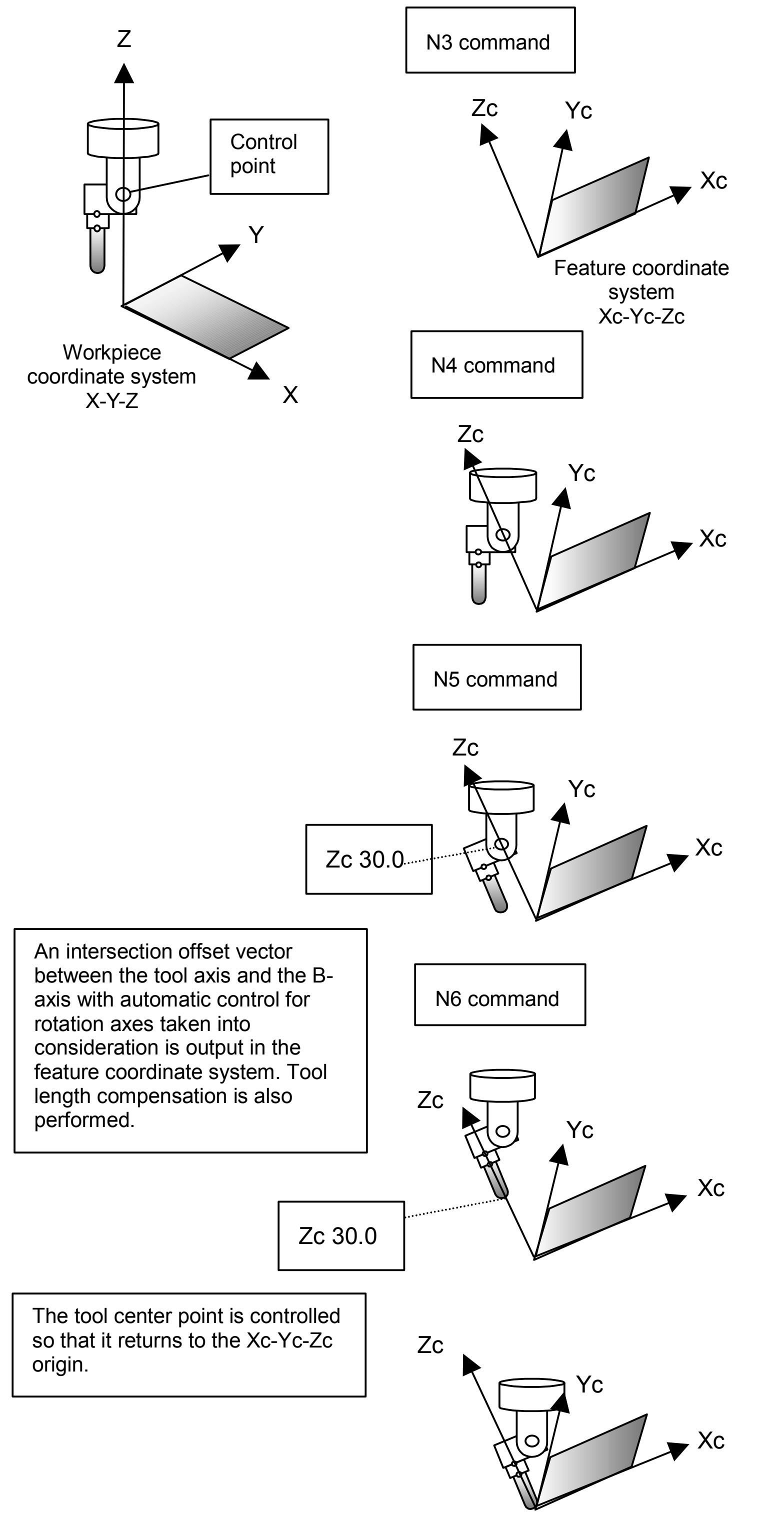


Fig. 21.2 (g) Tool axis direction control 2

Operation description 3: When no G43 (tool length compensation) command is specified or if no G53.1 (tool axis direction control) command is specified

Sample program 2 of O200 is equivalent to sample program 1 except that sample program 2 has no tool length compensation command (G43).

Example)

```
O200 (Sample Program2) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z0 F1000 ;
N5 G53.1 ;
N6 … ;
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
The case in which the axes cross one another and the case in which no axis crosses any other axis are described.
Fig. 21.2 (h) shows the behavior of the machine when it runs sample program 2.

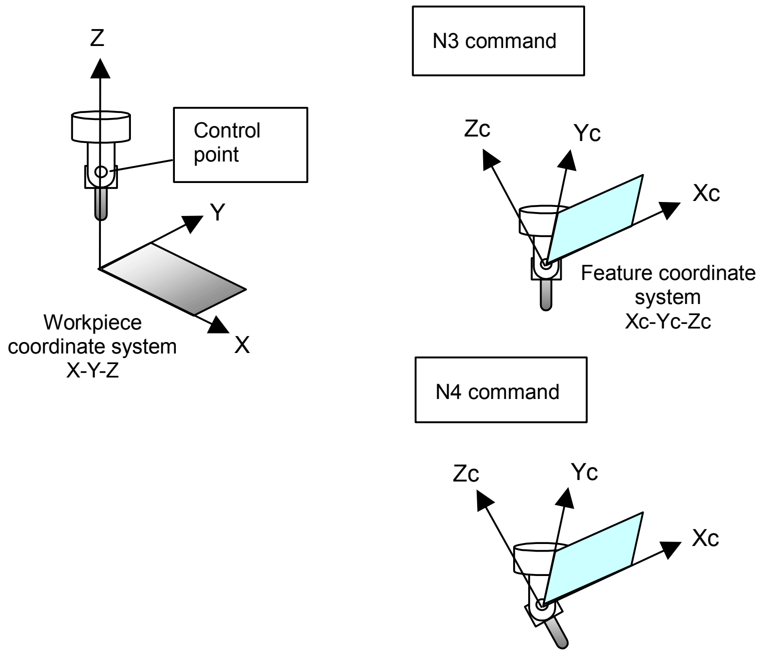
Sample program 3 of O300 is equivalent to sample program 1 except that sample program 3 has no tool axis direction control command (G53.1).

Example)

```
O300 (Sample Program3) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z0 F1000 ;
N5 G43 H01 ;
N6 … ;
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
The case in which the axes cross one another and the case in which no axis crosses any other axis are described.
Tool length compensation is applied in the +Z-axis direction of the feature coordinate system.
Fig. 21.2 (i) shows the behavior of the machine when it runs sample program 3.

- **Sample program 2 (with axes crossing one another)**



- **Sample program 2 (no axis crossing)**

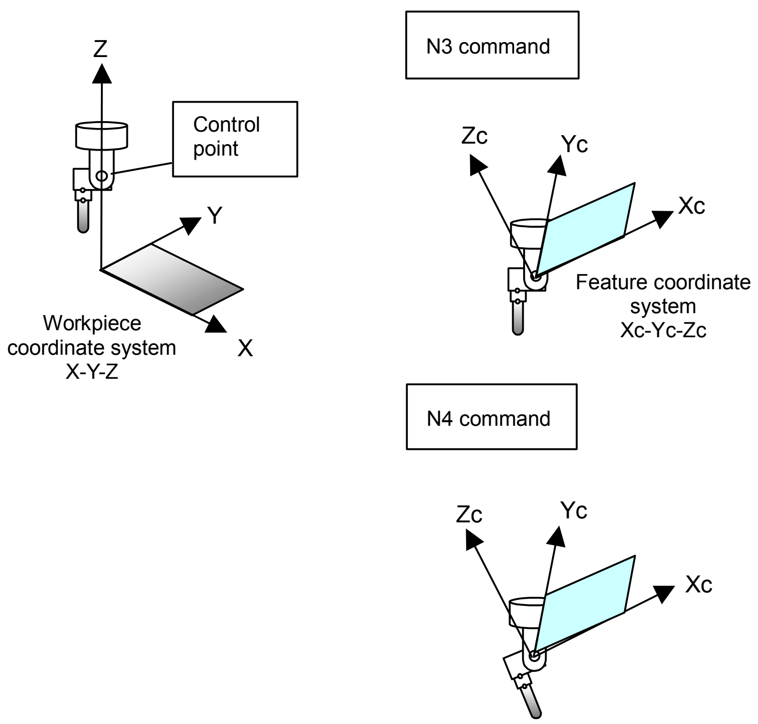
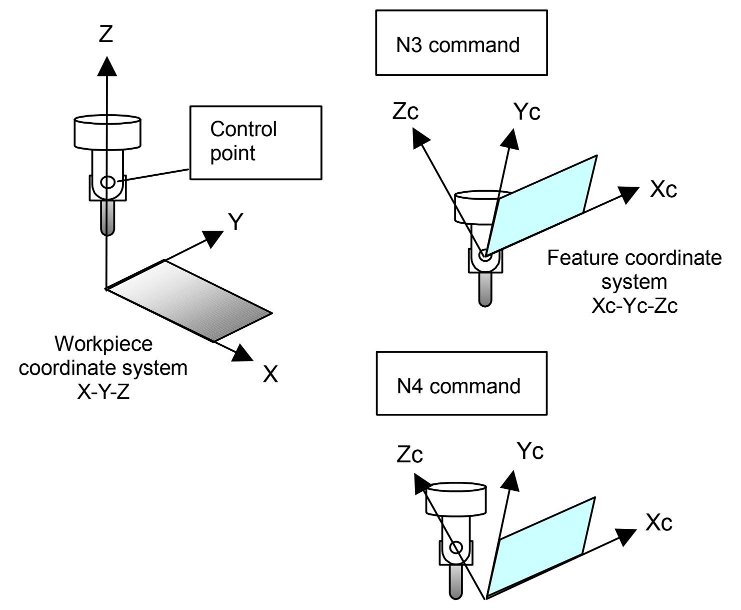


Fig. 21.2 (h) When the tool length compensation command is not specified

- **Sample program 3 (with axes crossing one another)**



- **Sample program 3 (no axis crossing)**

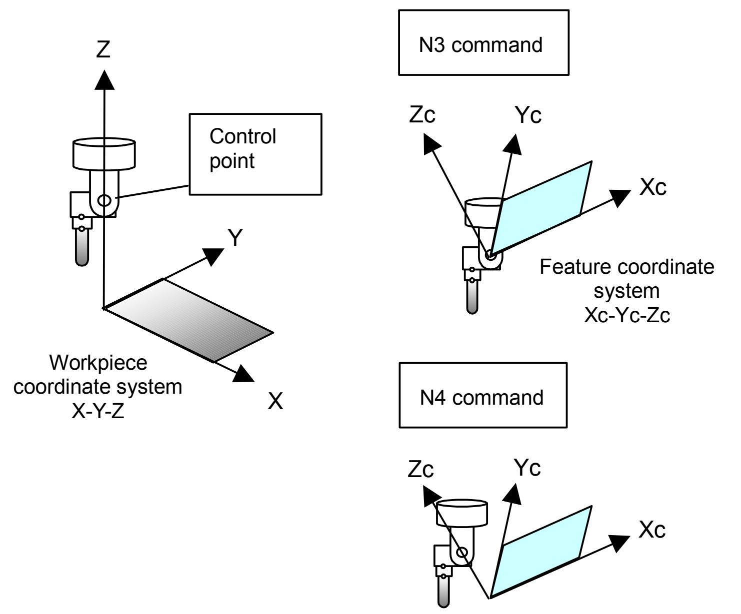


Fig. 21.2 (i) When the tool axis direction control command is not specified

- Mixed-type machine Basic operation

This function is also available for a mixed-type machine in which the tool head rotates on the tool rotation axis and the table rotates on the table rotation axis.

The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle.

Given the A-axis and B-axis shown in Fig. 21.2 (j), control is performed in such a way that the A-axis rotates until Zc comes in the X-Z plane and the B-axis is controlled so that the tool axis is oriented toward the +Z-axis direction of the feature coordinate system.

- **Tool axis direction control for a mixed-type machine**

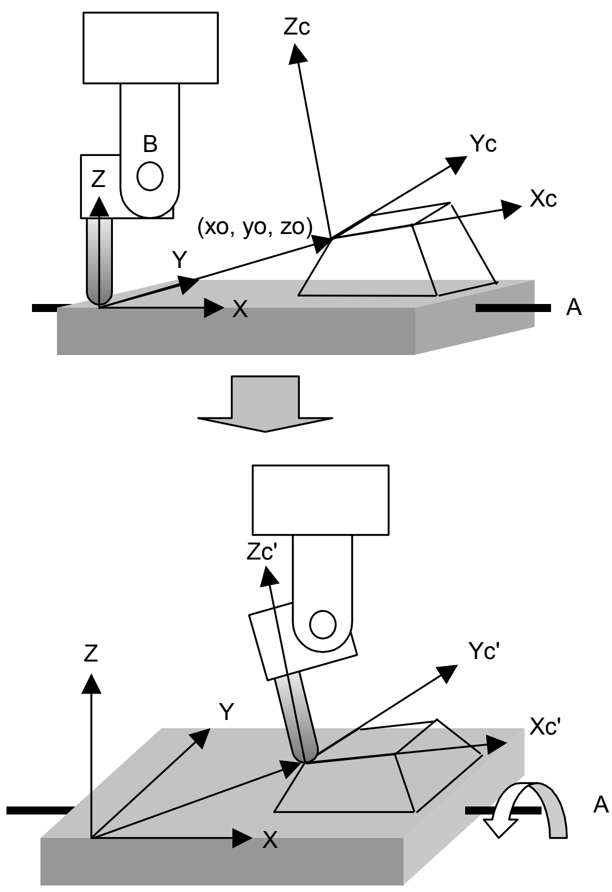


Fig. 21.2 (j) Mixed-type machine

- Feature coordinate system with the table rotated by G53.1 (tool axis direction control)

The mixed-type machine shown in Fig. 21.2 (j) is explained as an example.
If the table rotates by the tool axis direction control command (G53.1), the feature coordinate system (called the first feature coordinate system), which is set in the workpiece coordinate system by the feature coordinate system set command (G68.2), rotates as much as the table rotates.
The feature coordinate system that has rotated is called the second feature coordinate system.
Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. (See Fig. 21.2 (k).)
In the mixed-type machine, the specified feature coordinate system (the first feature coordinate system) may differ from the feature coordinate system to be used for machining (the second feature coordinate system).

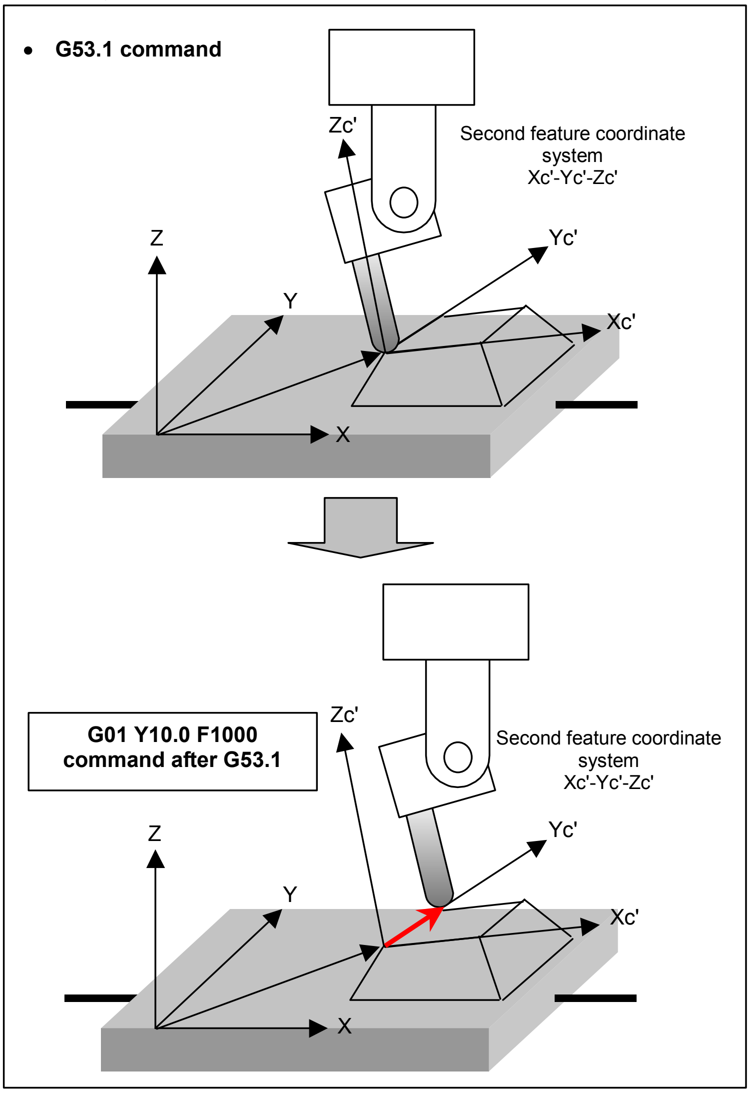


Fig. 21.2 (k) Resetting of the feature coordinate system

- Rotation direction of the table rotation axis

The mixed-type machine shown in Fig. 21.2 (j) is explained as an example.

Set parameter No.19684 to 1 if the rotation direction of the rotation table corresponding to the positive-direction move command is clockwise when viewed from the positive direction of the rotation center axis on which the table rotation axis rotates. If the rotation direction is counterclockwise, set parameter No.19684 to 0.

Let's take sample program 4 of O400 as an example, where the movement of the table is specified by G53.1.

If parameter No.19684 is set to 1, control is performed in such a way that the table is rotated to A-45.0.

If parameter No.19684 is set to 0, control is performed in such a way that the table is rotated to A45.0.

Example)

```
O400 (Sample Program4) ;
N1 G68.2 X100.0 Y100.0 Z0 I180.0 J45.0 K0 ;
N2 G53.1 ;
N3 … ;
```

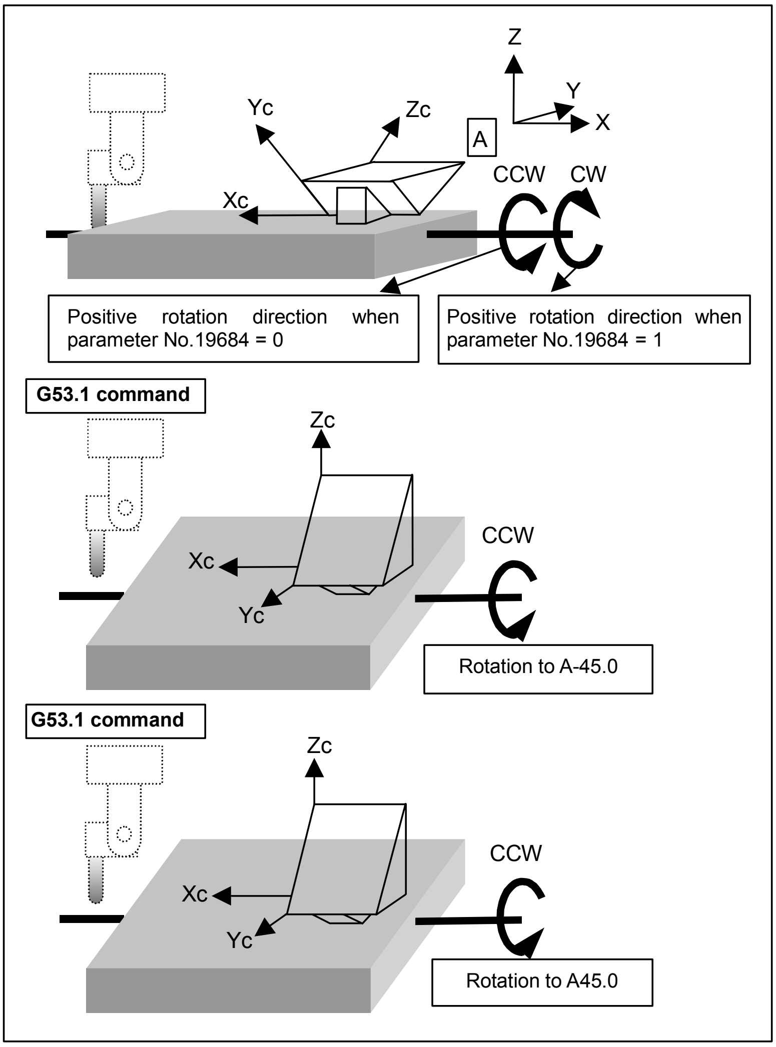


Fig. 21.2 (l) Rotation direction of the table rotation axis

- Table rotation type machine
Basic operation

This function is also usable for a table rotation type machine with two table rotation axes.
The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle.
Given the A-axis and C-axis shown in Fig. 21.2 (m), the A-axis and C-axis rotate until Zc comes in the X-Z plane and the tool axis is directed toward the +Z-axis direction of the feature coordinate system.

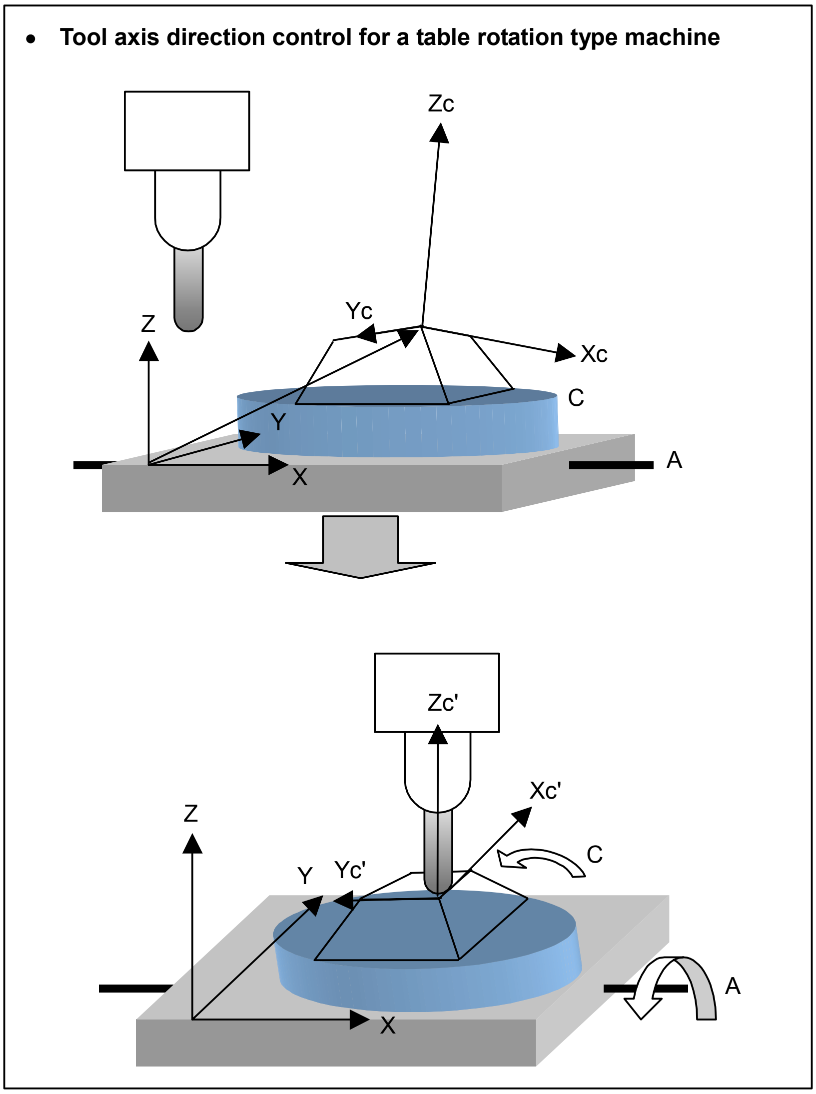


Fig. 21.2 (m) Table rotation type machine

- Feature coordinate system with the table rotated by G53.1 (tool axis direction control)

The table rotation type machine shown in Fig. 21.2 (m) is explained as an example.
If the table rotates by the tool axis direction control command (G53.1), the feature coordinate system (called the first feature coordinate system), which is set in the workpiece coordinate system by the feature coordinate system set command (G68.2), rotates as much as the table rotates.
The feature coordinate system that has rotated is called the second feature coordinate system.
Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. (See Fig. 21.2 (n).)
In the table rotation type machine, the specified feature coordinate system (the first feature coordinate system) may differ from the feature coordinate system to be used for machining (the second feature coordinate system).

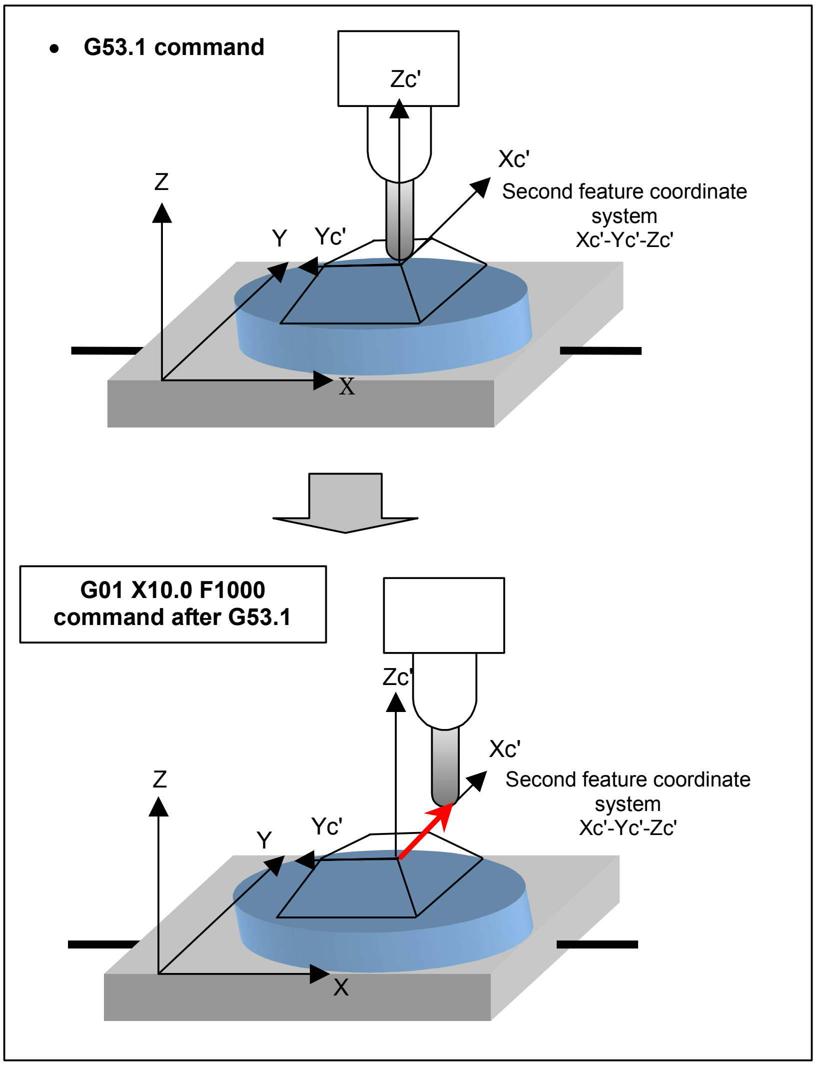


Fig. 21.2 (n) Resetting of the feature coordinate system

- Angle of the rotary axis

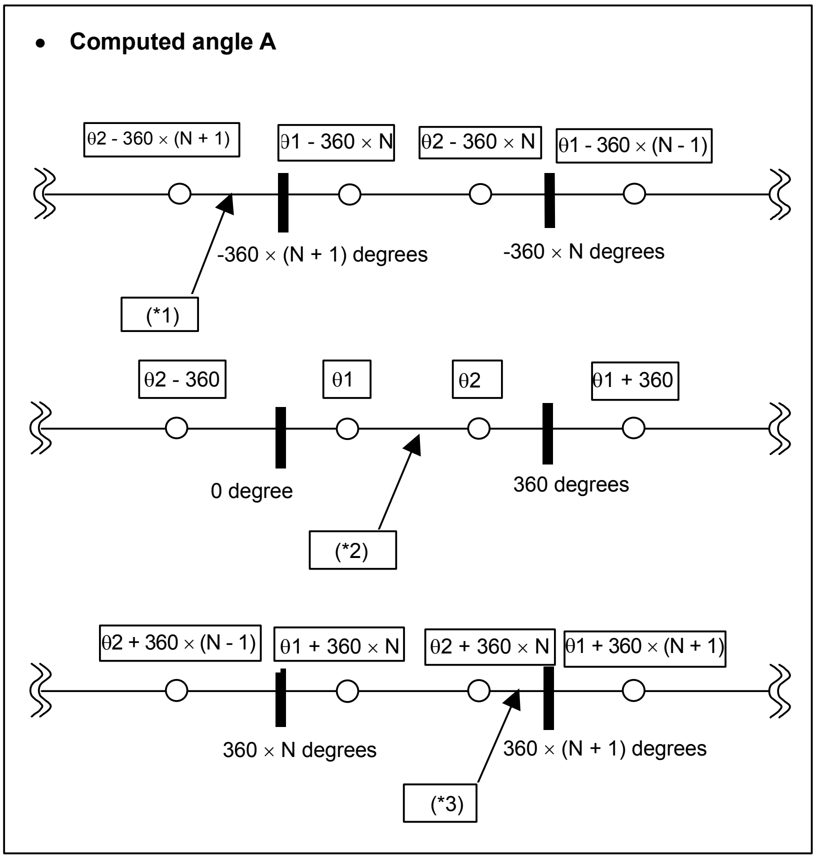
When tool axis direction control (G53.1) has been performed, more than two pairs of "computed angles" of the rotary axes usually exist.
The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the tool axis direction specified by G53.1.
The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.

"Output judgment conditions"

Tool rotation type or table rotation type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller. ↓ ↓ When the master axis moving angle is the same ↓ <2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller. ↓ ↓ When the slave axis moving angle is the same ↓ <3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓ <4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
Mixed-type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller. ↓ ↓ When the table moving angle is the same ↓ <2> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller. ↓ ↓ When the tool moving angle is the same ↓ <3> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓ <4> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

The process of judging whether the moving angle is smaller or larger as the output judgement condition is called "movement judgement."

The "movement judgement" process is explained below.
When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."
Usually, two pairs of "basic computed angles" exist.
For example, assume that the machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A $\theta 1$ degree; B $\phi 1$ degree)
(A $\theta 2$ degrees; B $\phi 2$ degrees) where $\theta 1 \leq \theta 2$ and $\phi 1 \leq \phi 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.
Based on the PA angle, the "movement judgement" process is done as follows.



"Movement judgment"

When the PA angle is (*1):
The output angle is: (A $\theta 2 - 360 \times (N + 1)$ degrees; B $\phi 2$ degrees).
Namely, $\theta 2 - 360 \times (N + 1)$ degrees is adopted that is nearer to the computed angle of A, and $\phi 2$, which is the same group as $\theta 2$, is adopted as the computed angle of B.

When the PA angle is (*2):
The output angle is: (A $\theta 1$ degrees; B $\phi 1$ degrees).
Namely, $\theta 1$ degrees is adopted that is nearer to the computed angle of A, and $\phi 1$, which is the same group as $\theta 1$, is adopted as the computed angle of B.

When the PA angle is (*3):
The output angle is: (A $\theta 2 + 360 \times N$ degrees; B $\phi 2$ degrees).
Namely, $\theta 2 + 360 \times N$ degrees is adopted that is nearer to the computed angle of A, and $\phi 2$, which is the same group as $\theta 2$. is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgement" is made for rotary axis B (slave) according to the "output judgment conditions."
If the "output angle" of rotary axis A is determined by the "movement judgement" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis B.
Similarly, if the "output angle" of rotary axis B is determined by the "movement judgement" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

⚠ CAUTION

1. To use the rotary axis roll-over function, set parameter No. 1260 (amount of rotary axis movement per rotation) to 360 degrees.
2. A stroke limit before movement is applied to the rotary axis subject to tool axis direction control.

The "output angle" is explained below using a tool rotation type machine as an example.
This example illustrates a machine having a "BC type tool axis Z."

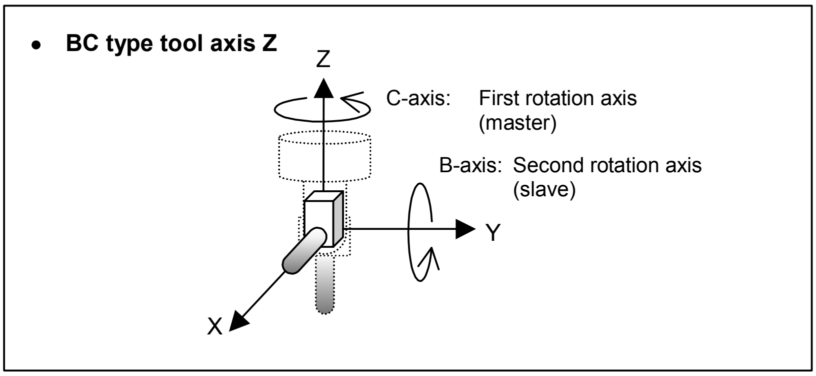


Fig. 21.2 (o) BC type tool axis Z

The following two pairs of "computed basic angles" exist that direct the tool axis toward the +X-axis direction.
(B 90 degrees; C 180 degrees)
(B 270 degrees; C 0 degree)

<1> When the current rotary axis angles are (B -70 degrees; C 30 degrees)
The "output angles" are (B -90 degrees; C 0 degree).
0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).

<2> When the current rotary axis angles are (B 80 degrees; C 500 degrees)
The "output angles" are (B 90 degrees; C 540 degrees).
540 degrees (180 degrees + 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.

<3> When the current rotary axis angles are (B 60 degrees; C 90 degrees)
The "output angles" are (B 90 degrees; C 180 degrees).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.

<4> When the current rotary axis angles are (B 180 degrees; C 90 degrees)
The "output angles" are (B 270 degrees; C 0 degree).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree.
That is, the pair is adopted whose C-axis angle is 0 degree and whose B-axis angle is 270 degrees.

When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.
In that case, the master axis does not move from the current angle.

An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

- **BC type tool axis Z**

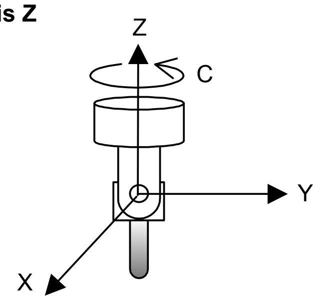


Fig. 21.2 (p) BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

Limitation

- Basic restrictions

The restrictions for this function are similar to those for the three-dimensional coordinate conversion function.

- Increment system

The same increment system must be used for the basic three axes used by this function.

- Rapid traverse command

The rapid traverse command must specify linear rapid traverse (parameter LRP (parameter No.1401#1) = 1).

- Feature coordinate system and three-dimensional coordinate conversion

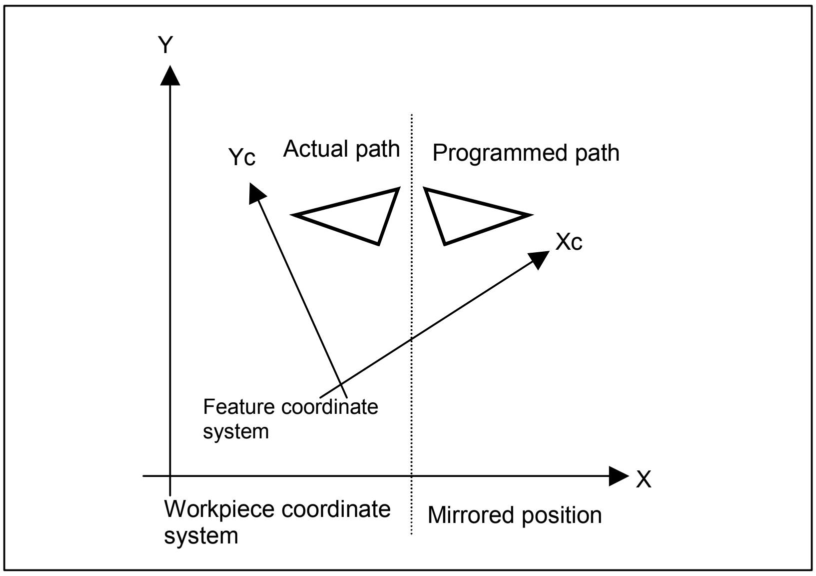
An alarm occurs if an attempt is made to set a feature coordinate system in another feature coordinate system.
An alarm also occurs if an attempt is made to set a new coordinate system by performing three-dimensional coordinate conversion in a feature coordinate system.

- Positioning in the machine coordinate system

Positioning commands in the machine coordinate system, such as G28, G30, and G53, operate in the machine coordinate system rather than in the feature coordinate system.

- External mirror image

If an attempt is made to use this function and the external mirror image function simultaneously, this function takes effect before the external mirror image function.



- Relationships with other modal commands

G41, G42, and G40 (cutter compensation), G43 and G49 (tool length compensation), G51.1 and G50.1 (programmable mirror image), and canned cycle commands must have nesting relationships with G68.2.
In other words, first issue G68.2 when the modes mentioned above are off, then turn the modes on and off, and then issue G69.

- Parallel axis control

When a parking signal is applied to an axis during parallel axis control, conversion to a feature coordinate system occurs for another axis if a move command is issued for that another axis. For this reason, an axis may move even if a parking signal has been applied to it.

- Specifiable G codes

The G codes that can be specified in the tilted working plane command mode are listed below.
Specifying a G code other than these codes results in alarm PS5462.

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation / helical interpolation (G02/G03)
- Dwell (G04)
- Programmable data input (G10)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Automatic return to reference position (G28)
- Movement from reference position (G29)
- 2nd, 3rd and 4th reference position return (G30)
- Cutter compensation : cancel (G40)
- Cutter or tool nose radius compensation /
  Three-dimensional cutter compensation (G41/G42)
- Tool length compensation + (G43)
- Tool length compensation - (G44)
- Tool length compensation cancel (G49)
- Machine coordinate system setting (G53)
- Tool axis direction control (G53.1)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Absolute programming (G90)
- Incremental programming (G91)
- Canned cycle for drilling (G73, G74, G76, G80 to G89)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Tool offset increase (G45)
- Tool offset decrease (G46)
- Tool offset double increase (G47)
- Tool offset double decrease (G48)
- Programmable mirror image (G50.1/G51.1)
- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69)
- Feed per minute (G94)
- Feed per revolution (G94)

T

- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69.1)
- Feed per minute (G98 (G94))
- Feed per revolution (G99 (G94))

- Modal G codes that allow specification of a tilted working plane command

A tilted working plane command can be specified in the modal G code states listed below.
In a modal state other than the following modal G codes, specifying the tilted working plane command results in alarm PS5462:

- Modal G codes included in "Specifiable G codes" described previously
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Stored stroke check function (G22/G23)
- Scaling cancel (G50)
- Polygon turning cancel (G50.2)
- Workpiece coordinate system 1 selection (G54 to G59)
- Exact stop mode (G61)
- Automatic corner override (G62)
- Tapping mode (G63)
- Cutting mode (G64)
- Inverse time feed (G93)
- Constant surface speed control cancel (G97)

M

- Polar coordinate interpolation mode cancel (G113)

T

- Programmable mirror image cancel (G50.1)
- Mirror image for double turret off/balanced cutting mode cancel (G69)

21.3 INCLINED ROTARY AXIS CONTROL

Overview

The conventional tilted working plane command / tool center point control function for 5-axis machining / cutter compensation for 5-axis machining / manual handle feed for 5-axis machining can be used only for those machines whose tool rotation axis or table rotation axis is parallel to the basic axis of the basic coordinate system. (See Fig. 21.3 (a).)

Inclined rotary axis control is intended to allow the tilted working plane command / tool center point control function for 5-axis machining / cutter compensation for 5-axis machining / manual handle feed for 5-axis machining to be applied even when the tool rotation axis or table rotation axis is inclined with respect to the X-Z plane, a Y-Z plane or a Z-X plane of the machine coordinate system.

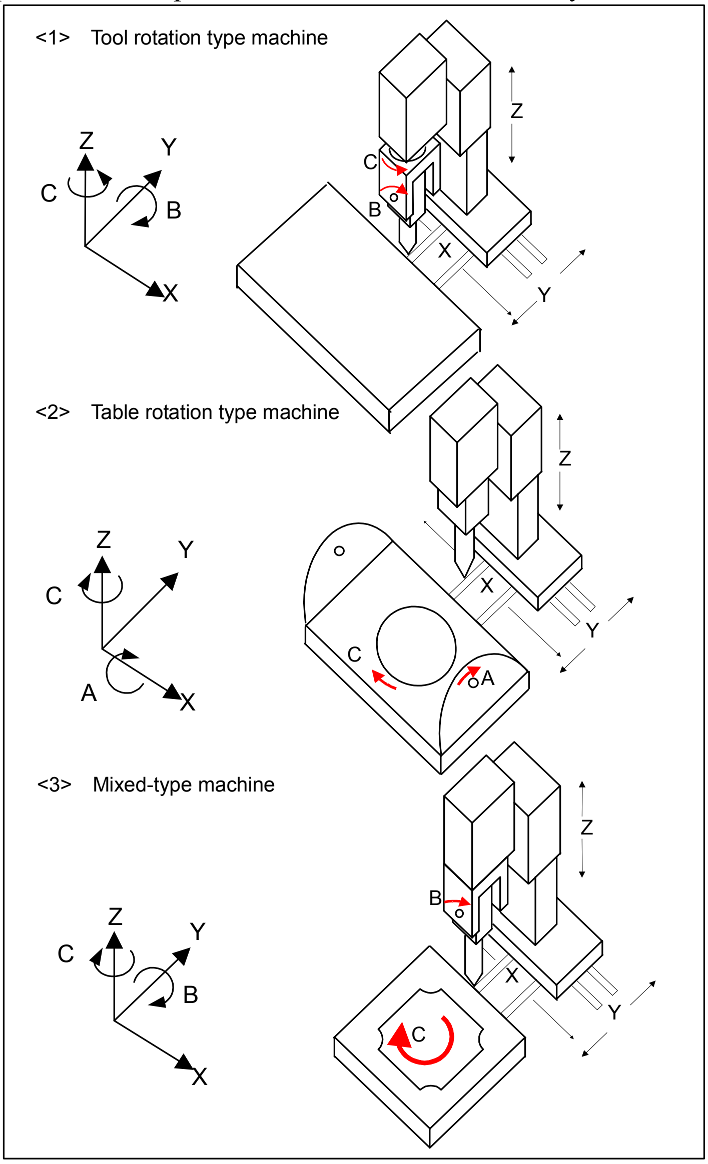


Fig. 21.3 (a) Three types of 5-axis machine

An example of a tool rotation type machine is explained below. (See Fig. 21.3 (b).)
The machine shown in Fig. 21.3 (b) has rotary axis B (master) that turns around the Y-axis and rotary axis C (slave) whose Y-axis is inclined at an angle of 45 degrees on the Y-Z plane.
The tilted working plane command / tool center point control function for 5-axis machining / cutter compensation for 5-axis machining / manual handle feed for 5-axis machining can be used even for the machine configuration shown in Fig. 21.3 (b).

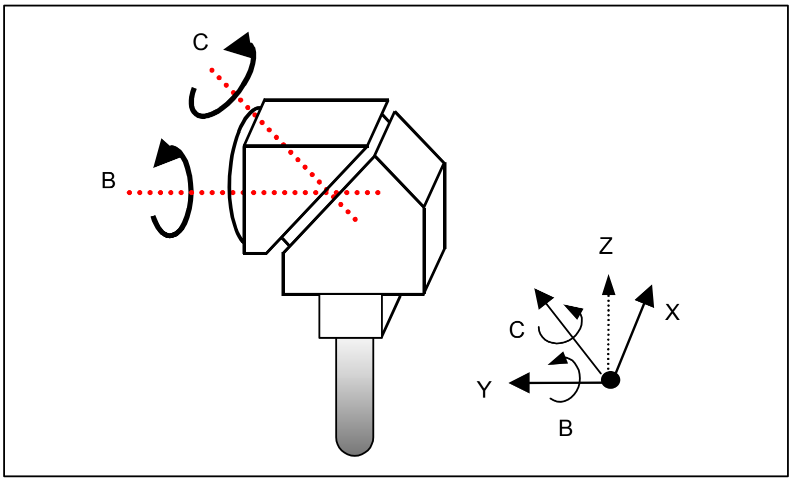


Fig. 21.3 (b) Tool rotation type machine

An example of a table rotation type machine is explained below. (See Fig. 21.3 (c).)
The machine shown in Fig. 21.3 (c) has rotary axis B (master) whose Y-axis is inclined at an angle of -45 degrees on the Y-Z plane and rotary axis C (slave) that turns around the Z-axis.
The tilted working plane command / tool center point control function for 5-axis machining / cutter compensation for 5-axis machining / manual handle feed for 5-axis machining can be used even for the machine configuration shown in Fig. 21.3 (c).

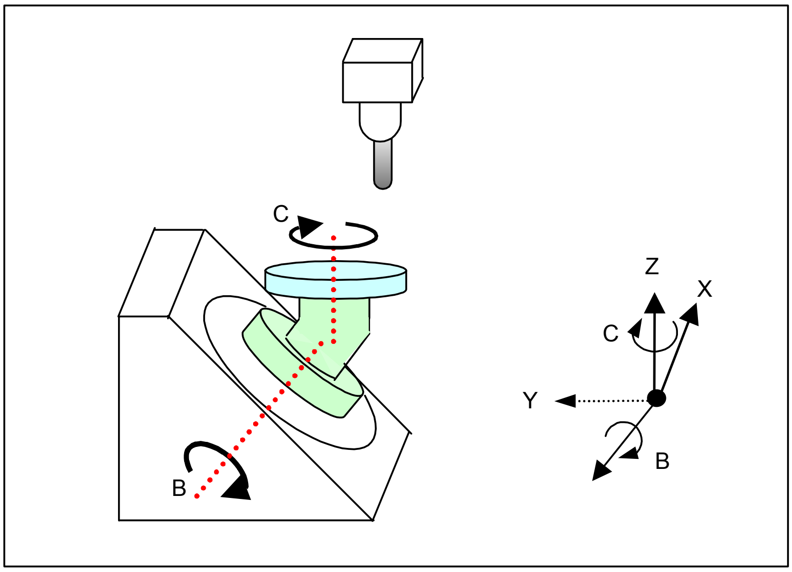


Fig. 21.3 (c) Table rotation type machine

An example of a mixed-type machine is explained below. (See Fig. 21.3 (d).)
The machine shown in Fig. 21.3 (d) has table rotation axis B whose Y-axis is inclined at an angle of -45 degrees on the Y-Z plane and tool rotation axis C that turns around the Z-axis.
The tilted working plane command / tool center point control function for 5-axis machining / cutter compensation for 5-axis machining / manual handle feed for 5-axis machining can be used even for the machine configuration shown in Fig. 21.3 (d).

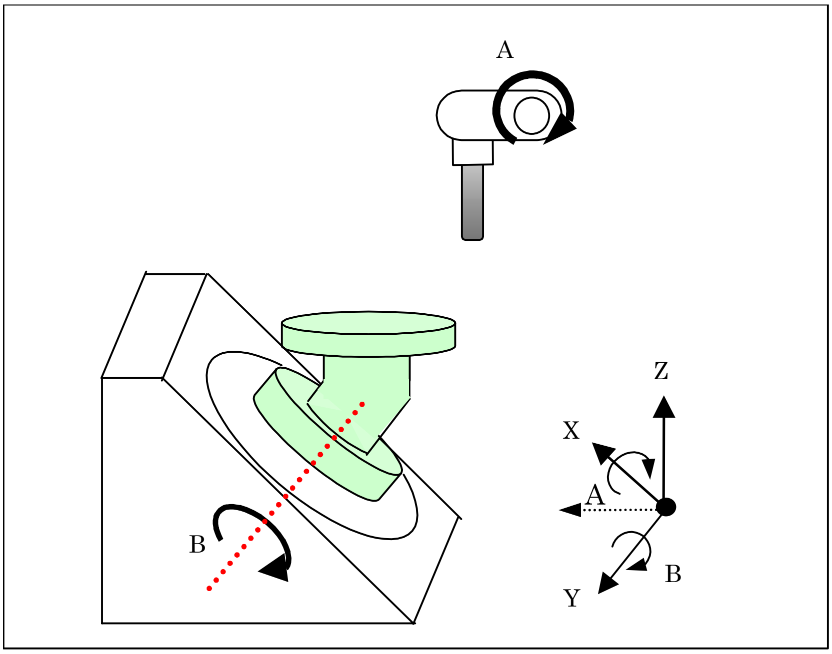


Fig. 21.3 (d) Mixed-type machine

Format and operation

The operation of the tilted working plane command / tool center point control function for 5-axis machining / cutter compensation for 5-axis machining / manual handle feed for 5-axis machining during the inclined rotary axis control is similar to the operation performed when the inclined rotary axis control is not in use.
For details, see the description of each function.

Limitation

The operation of the tilted working plane command / tool center point control function for 5-axis machining / cutter compensation for 5-axis machining / manual handle feed for 5-axis machining during the inclined rotary axis control is similar to the operation performed when the inclined rotary axis control is not in use.
For details, see the description of each function.

21.4 CUTTER COMPENSATION FOR 5-AXIS MACHINING

Overview

For machines having multiple rotary axes for freely controlling the orientation of a tool axis, this function calculates a tool vector from the positions of these rotary axes. The function then calculates a compensation vector in a plane (compensation plane) perpendicular to the tool vector and performs three-dimensional cutter compensation.

- Machine configuration

This function is applicable to the following machine configurations:

<1> Tool rotation type machine controlled with two tool rotation axes

<2> Table rotation type machine controlled with two table rotation axes

<3> Mixed-type machine controlled with one tool rotation axis and one table rotation axis

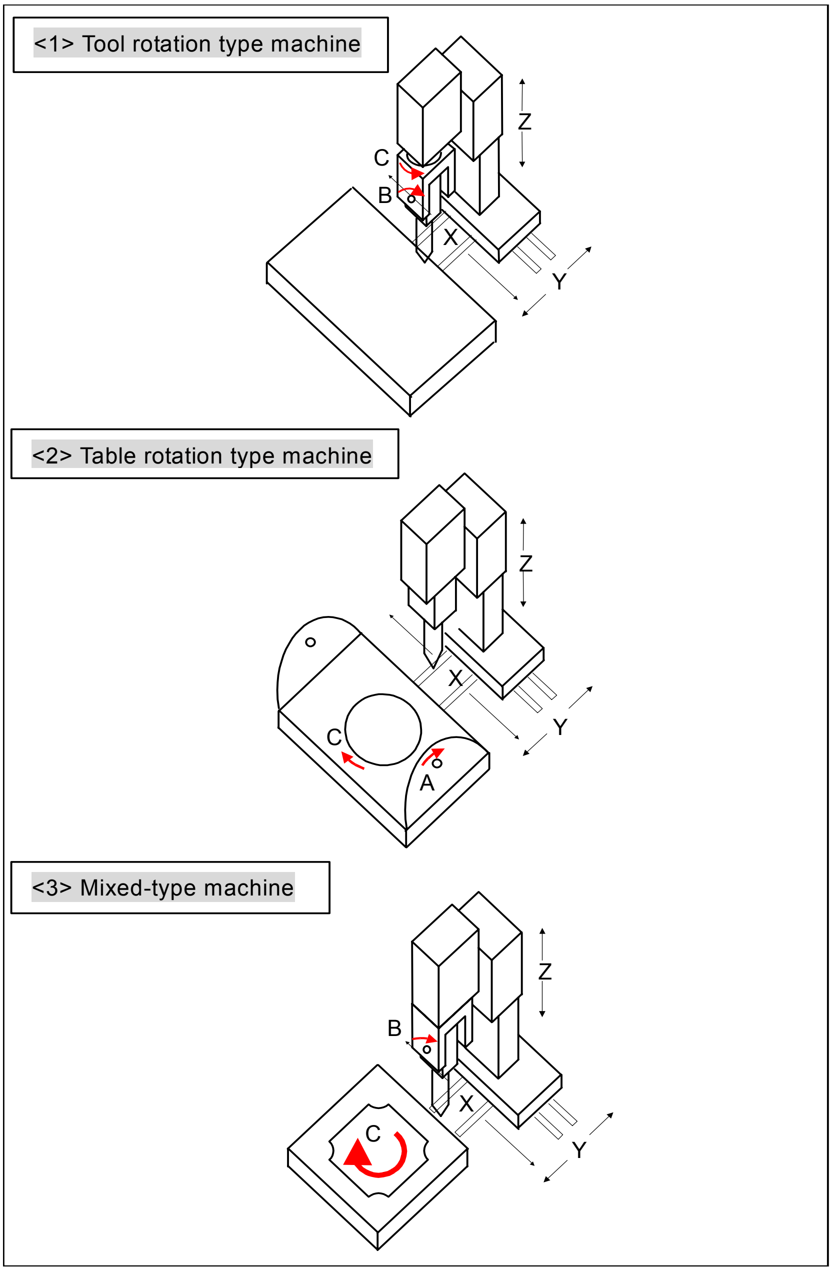


Fig. 21.4 (a) Three types of machine configuration

The coordinate system in which to execute a program for cutter compensation for 5-axis machining is called a programming coordinate system.
If, in a 5-axis machine having a table rotation axis, cutter compensation for 5-axis machining (tool side offset) is to be performed, the table coordinate system (coordinate system fixed to the table) can be selected as a programming coordinate system.
Alternatively, the workpiece coordinate system fixed to the machine coordinate system can be selected as a programming coordinate system.
There are two types of cutter compensation for 5-axis machining (tool side offset), which differ in the method of specifying the tool axis direction.

(1) Type 1

Specify the block end position (for example, A, B, C) of the rotation axis. The CNC performs cutter compensation on the plane perpendicular to the tool axis direction calculated from the specified position on the rotation axis.

(2) Type 2

Instead of specifying a position on the rotation axis, specify the tool axis direction (I, J, K) at the block end point as viewed from the table coordinate system. The CNC calculates the end position on the rotation axis so that the tool points to the specified direction at the end point, and performs cutter compensation on the plane perpendicular to the tool axis direction calculated from the position on the rotation axis.

By using type 2, it is possible to perform the same machining with the same program regardless of the machine configuration of the 5-axis machine (tool rotation type, table rotation type, or mixed-type).

Same program

```
O0001
  :
G41.6   D1
G01 X100. I1.0 K1.0 F1000
  :
```

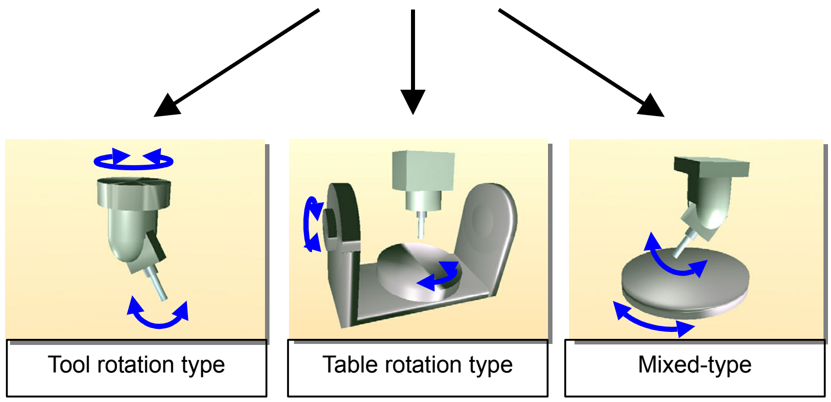


Fig. 21.4 (b) Type 2 Program

21.4.1 Cutter Compensation in Tool Rotation Type Machine

Overview

In a 5-axis machine having two tool rotation axes as shown in Fig. 21.4.1 (a), this function can perform cutter compensation.
Shown below is a 5-axis machine that has tool rotation axis B on the Y-axis and tool rotation axis C on the Z-axis.
This machine configuration is used as a sample configuration in the following explanation unless otherwise noted.

Cutter compensation in tool rotation machines is classified into two types according to the way of machining: tool side offset and leading edge offset.

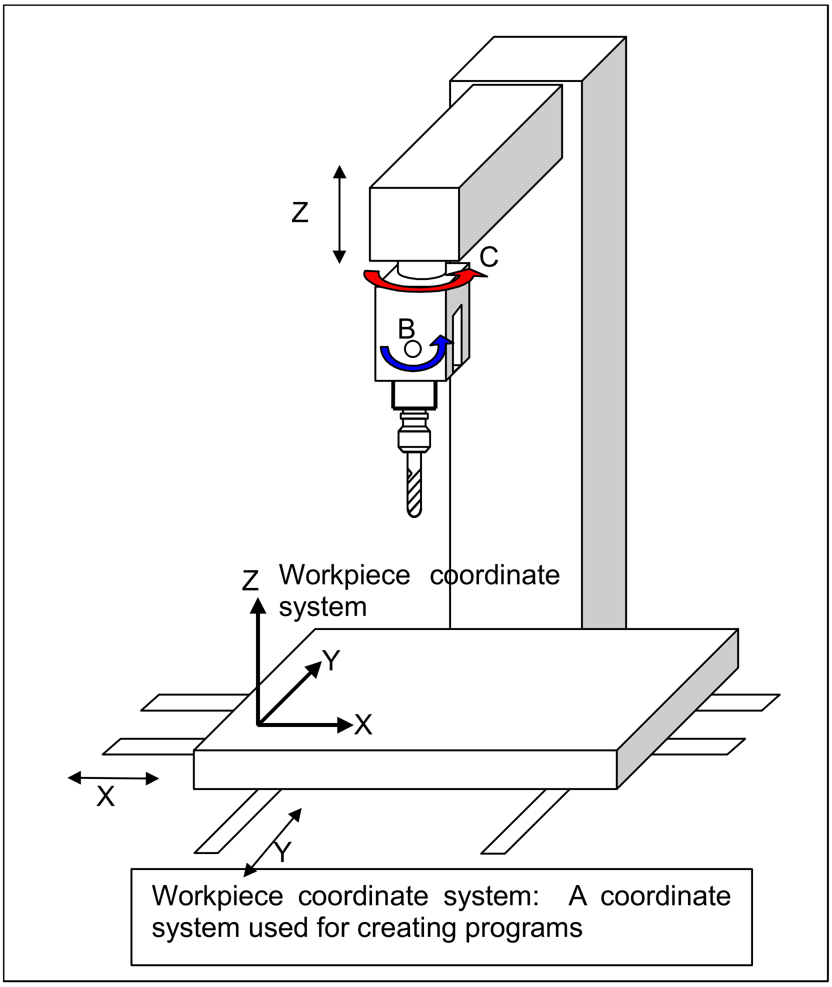


Fig. 21.4.1 (a) Machine having two tool rotation axes

21.4.1.1 Tool side offset

Overview

This type of cutter compensation performs three-dimensional compensation in a plane (compensation plane) perpendicular to the tool vector.

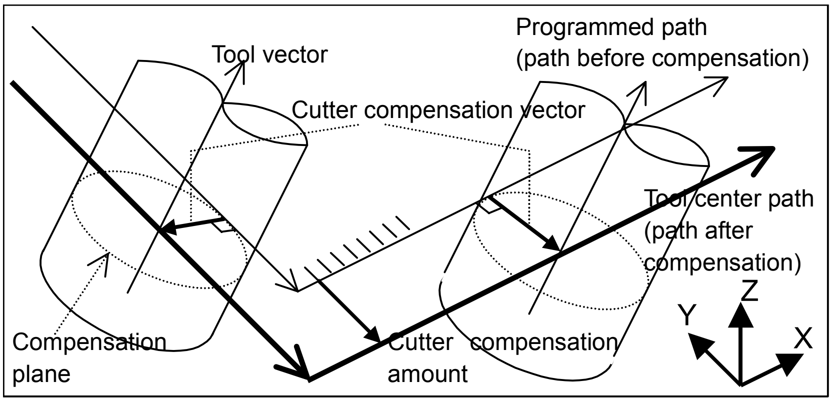


Fig. 21.4.1.1 (a) Tool side offset

Format

- Tool side offset (type 2)

G41.2 (or G42.2) IP_ D_ ;
IP_ I_ J_ K_ ;
:

G41.2:	Cutter compensation left (group 07)
G42.2:	Cutter compensation right (group 07)
IP_:	Value specified for axis movement (including rotary axis)
D_:	Code specifying the cutter compensation amount (1 to 3 digits)

- Tool side offset (type 1)

G41.6 (or G42.6) IP_ I_ J_ K_ D_ Q_ ;
IP_ I_ J_ K_ ;
:

G41.2:	Cutter compensation left (group 07)
G42.2:	Cutter compensation right (group 07)
IP_:	Value specified for axis movement (including rotary axis)
D_:	Code specifying the cutter compensation amount (1 to 3 digits)
Q_:	Tool’s angle of gradient (in degrees)
I_ J_ K_:	Tool axis direction at the block end point as viewed from the programming coordinate system

For type 2, do not specify a rotation axis but specify the direction at the tool end point as viewed from the programming coordinate system (workpiece coordinate system), with I, J, and K. Specifying a rotation axis causes alarm PS5460 to be generated.

> **NOTE**
> When a movement perpendicular to the next movement (bit 1 (SUV) of parameter No. 5003 is set to 1) is specified as the operation performed at the time of startup or cancellation, a move command such as X_ Y_ Z_ must not be specified in the G41.2 and G42.2 block.

The following are the notes on type 2.

> **NOTE**
> 1 If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
> 2 In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
> 3 If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
> 4 If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. 1260 (mount of travel per rotation about the rotation axis).
> 5 Only tool side offset provides type 2 commands. Leading edge offset, described later, does not provide type 2 commands.

- Canceling the tool side offset

> **G40 IP_ ;**
> G40: Cancellation of cutter compensation (group 07)
> IP_: Value specified for axis movement

Explanation

- Tool's angle of gradient in type 2

For type 2 of cutter compensation for 5-axis machining, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Fig. 21.4.1.1 (b).)

Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command.

If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary.

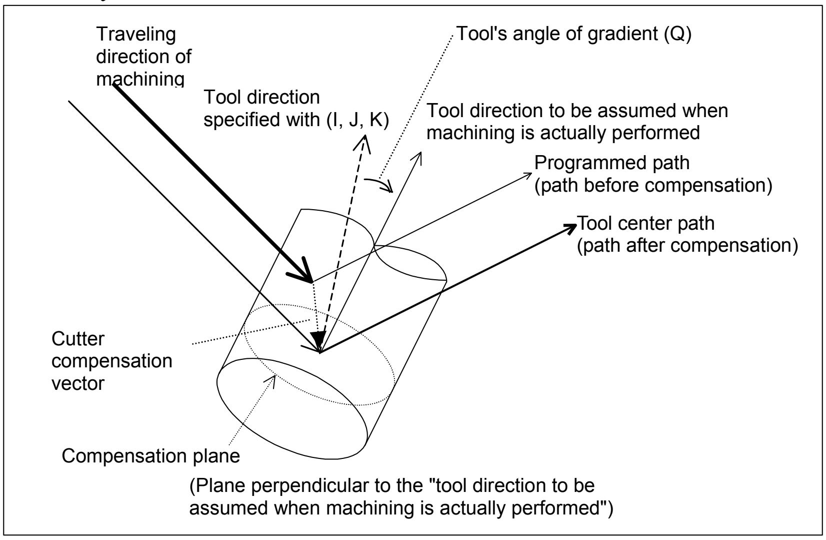


Fig. 21.4.1.1 (b) Tool's angle of gradient in type 2

(Example)

To perform machining by inclining the tool's traveling direction twice, issue a command such as the following:

G41.6 I_ J_ K_ H_ Q2.0

- Operation at startup and cancellation

<1> Type A

The tool is moved in the same way as for cutter compensation as shown below.

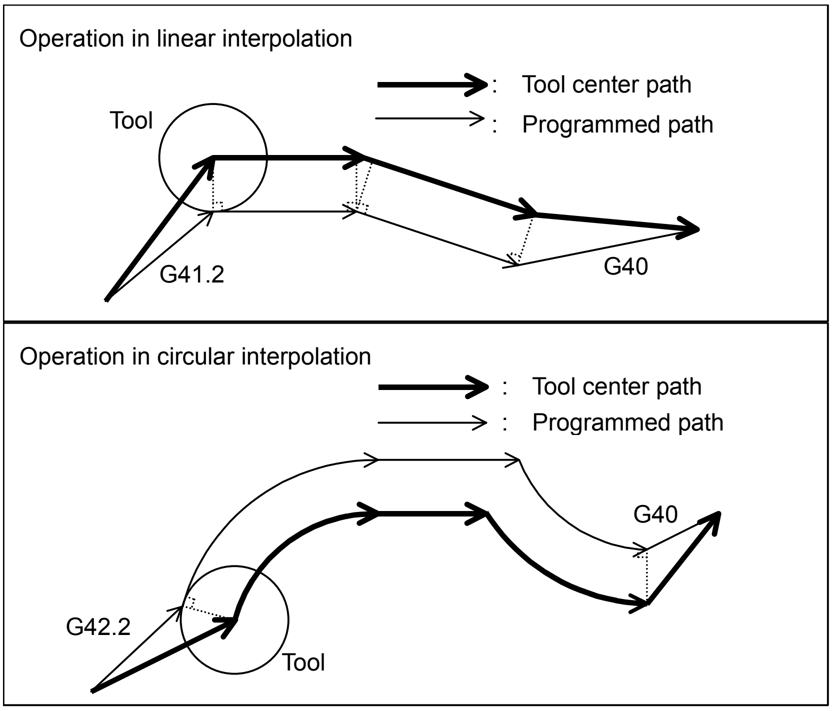


Fig. 21.4.1.1 (c) Operation at startup and cancellation (type A)

<2> Type B

The tool is moved in the same way as for cutter compensation as shown below.

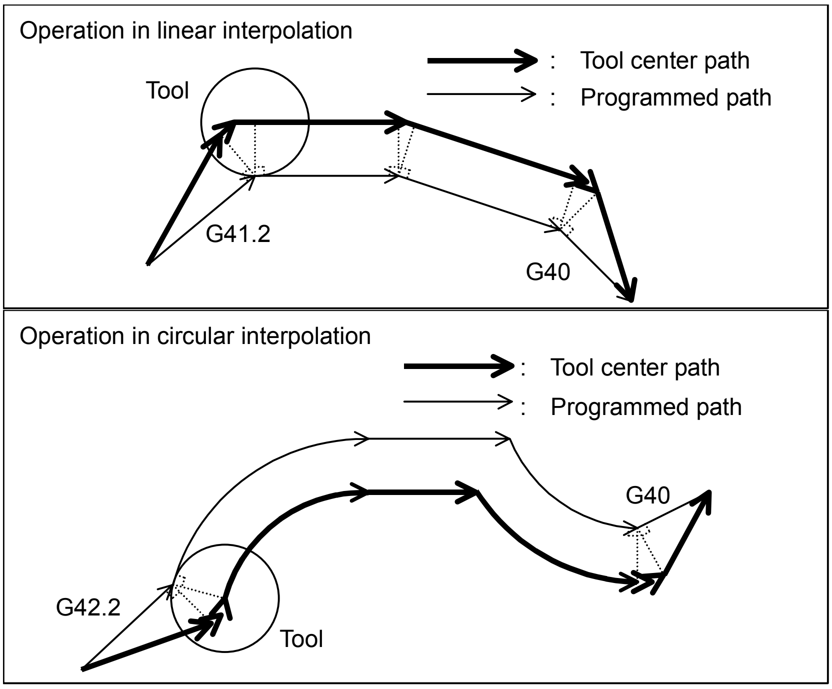


Fig. 21.4.1.1 (d) Operation at startup and cancellation (type B)

<3> Movement perpendicular to the next movement
When G41.2, G42.2, or G40 is specified, a block that moves the tool linearly by the amount of cutter compensation in a direction perpendicular to the movement direction of the next block is inserted as shown below.

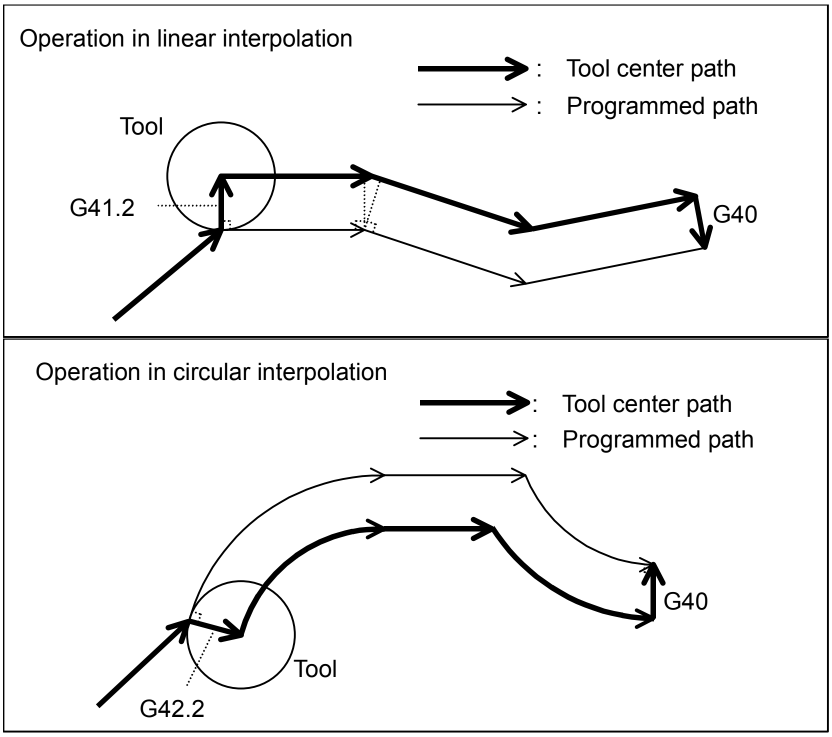


Fig. 21.4.1.1 (e) Operation at startup and cancellation (In a direction perpendicular to the next movement)

> **NOTE**
> When the movement direction is perpendicular to the next movement (bit 1 (SUV) of parameter No. 5003 is set to 1), the following conditions must always be satisfied at startup and cancellation:
> 1 A block specifying G40, G41.2, or G42.2 must be in the G00 or G01 mode.
> 2 A block specifying G40, G41.2, or G42.2 must contain no move command.
> 3 The block next to a block specifying G41.2 or G42.2 must contain move command G00, G01, G02, or G03.

- Operation during compensation

Operations such as change of the offset direction and offset value, retention of a vector, and interference checks are performed in the same way as for cutter compensation. However, G39 (corner rounding) cannot be specified. So, note the following:

<1> When the tool center path goes outside the programmed path at a corner, a linear movement takes place at the corner without inserting an arc. When the tool center path goes inside the programmed path, nothing is inserted.

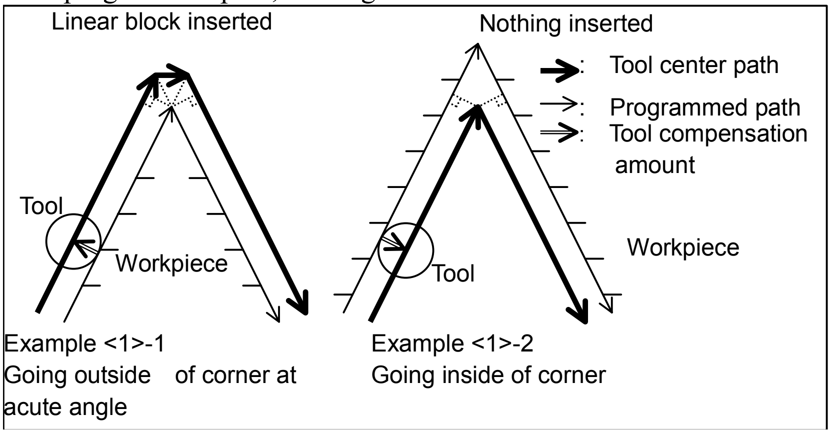


Fig. 21.4.1.1 (f) Operation during compensation <1>-1 and <1>-2

In the above examples, the terms "inside" and "outside" denote how the tool center path is positioned with respect to the programmed path. In the figure below, example <1>-3 shows the same relationship between the tool center path and programmed path as example <1>-1 and indicates that the tool center path is outside the programmed path; example <1>-4 shows the same relationship as example <1>-2 and indicates that the tool center path is inside the programmed path.

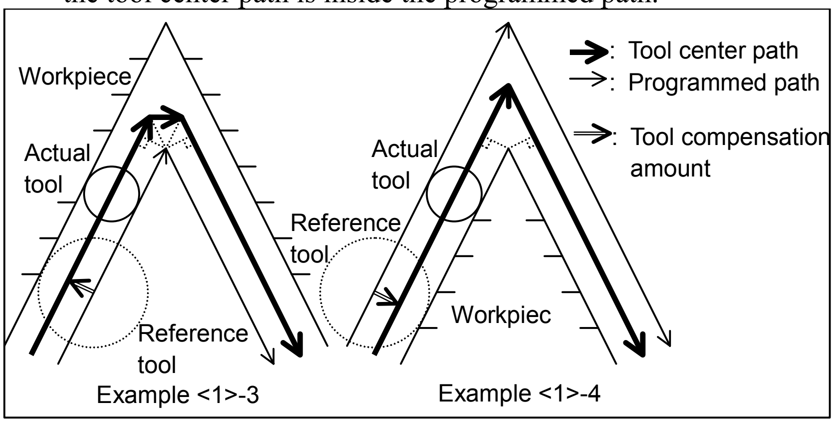


Fig. 21.4.1.1 (g) Operation during compensation <1>-3 and <1>-4

<2> When the tool moves at a corner, the feedrate of the previous block is used if the corner is positioned before a single-block stop point; if the corner is after a single-block stop point, the feedrate of the next block is used.

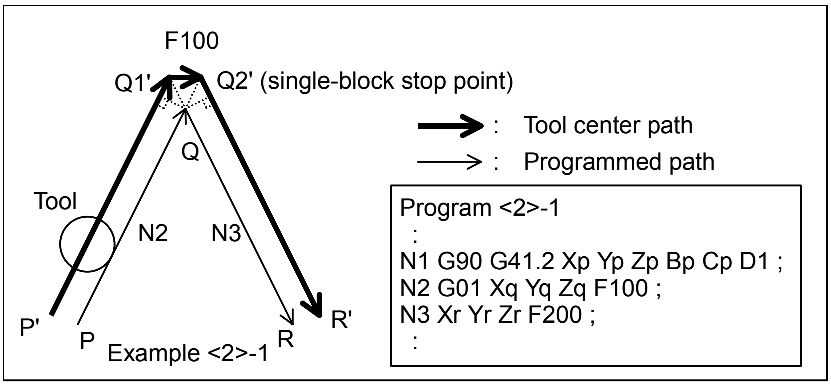


Fig. 21.4.1.1 (h) Operation during compensation <2>

In the above example, the single-block stop point of N2 is Q2', so the feedrates along paths P'-Q1' and Q1'-Q2' are both F100.

<3> When a command that makes the tool retrace the path of the previous block is specified, the tool path can match the locus of the previous block by changing the G code to change the offset direction. If the G code is left unchanged, the operation shown in example <3>-2 results:

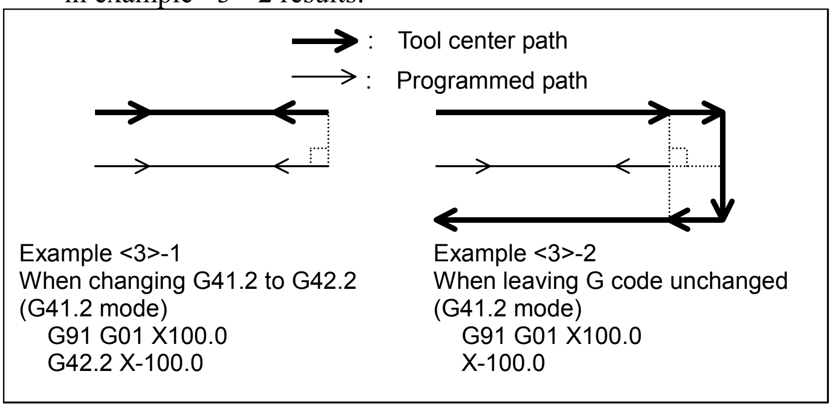


Fig. 21.4.1.1 (i) Operation during compensation <3>

- Interference check when the compensation plane changes

An interference check is made when the compensation plane (a plane perpendicular to the tool vector) has changed.

Example:

If the following program is executed, an alarm PS0041 (overcutting due to cutter or tool nose radius compensation) is issued at N4:

```
O100 F3000
N1 G90 G00 X0 Y0 Z0 A-46 C180
N2 G41.2 D1
N3 G01 X100
N4 Y-200 Z-200
N5 A45
N6 Y-400 Z0
N7 X0
N8 Y-200 Z-200
N9 A-46
N10 Y0 Z0
N11 G40
M30
```

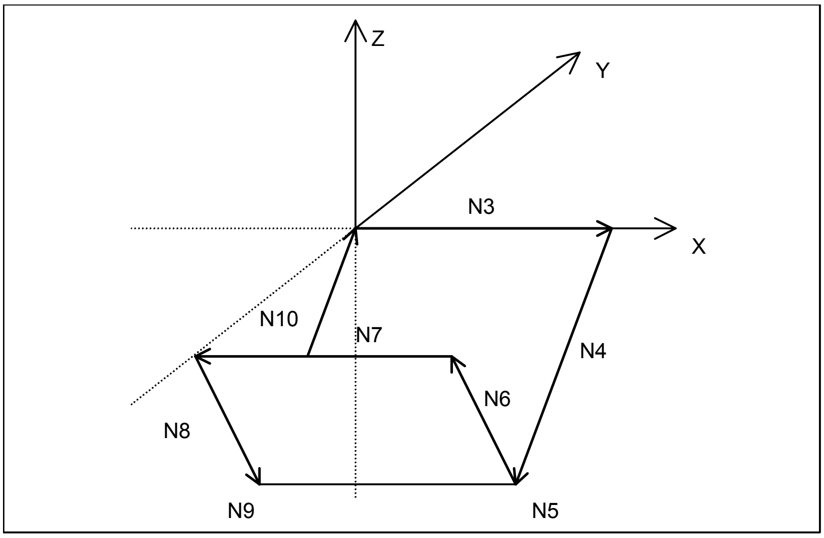


Fig. 21.4.1.1 (j) Conceptual diagram

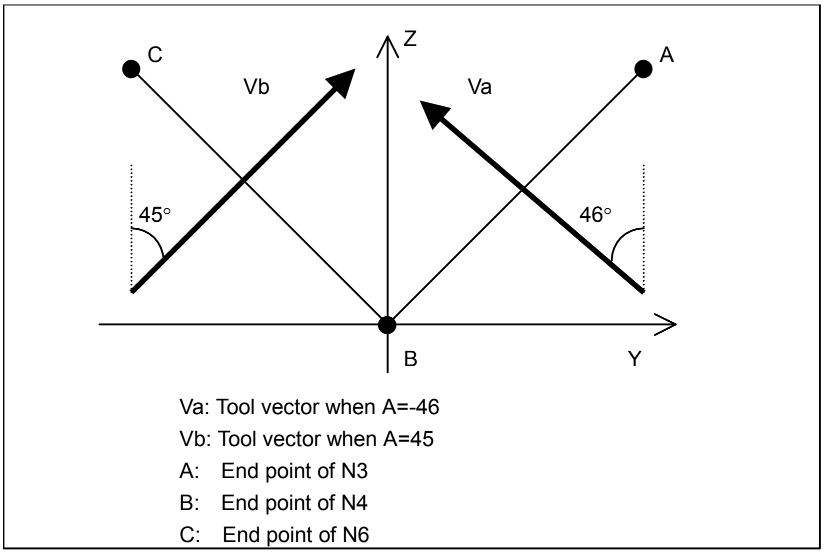


Fig. 21.4.1.1 (k) Tool vector

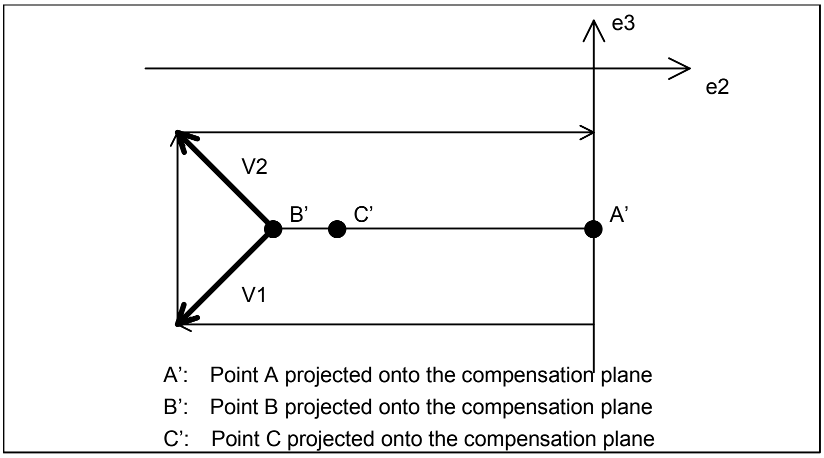


Fig. 21.4.1.1 (l) Compensation vector at the end point (point B) of N4 (in the compensation plane)

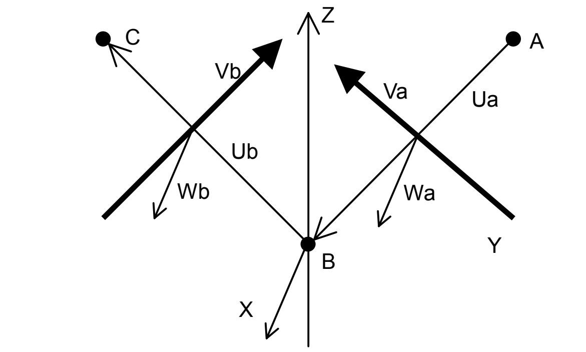
The movement direction of A'B' is opposite to that of B'C', so two compensation vectors V1 and V2 are produced at point B' (the end point of N4). In such a case, there is a possibility of overcutting, so an alarm PS0041 is issued at N4.

<1> Conditions for issuing the interference alarm

Suppose that a move command for a rotary axis makes the tool vector change significantly from one block to another. In this case, an interference alarm is assumed because compensation vectors are regarded as being generated in wrong directions when the path angle difference in the compensation plane is large, even though the angle difference between the directions of compensation vectors to be generated by those blocks is small.

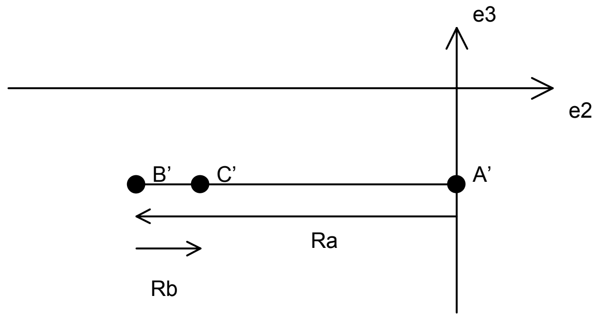
Here, the compensation plane is perpendicular to the tool direction (Va in the figure below) of the first of the two blocks.

Specifically, the conditions listed below are used for issuing the alarm.



Ua: Vector AB
Ub: Vector BC
Va: Tool vector between A and B
Vb: Tool vector between B and C
Wa: Va × Ua
Wb: Vb × Ub
(Here, × represents an outer product operator.)

Fig. 21.4.1.1 (m) Conceptual diagram



A': Point A projected onto the compensation plane
B': Point B projected onto the compensation plane
C': Point C projected onto the compensation plane
Ra: Vector A'B'
Rb: Vector B'C'

Fig. 21.4.1.1 (n) Programmed path before and after the end point (point B) of N4 (in the compensation plane)

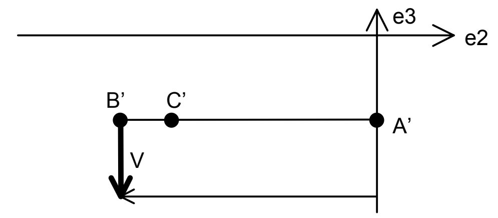
When all the following conditions are satisfied, an alarm PS0041) is issued:

(1) The tool vector changes significantly.
α: Angle for determination set in parameter No. 19635 (The default is 45°.)
$(Va,Vb) \leq \cos(\alpha)$(where, (Va,Vb) means an inner product.)

(2) The difference between the directions of the compensation vectors to be generated is small.
Wa: Direction of a compensation vector to be generated by block AB
Wb: Direction of a compensation vector to be generated by block BC
$Wa = Va \times Ua$
$Wb = Vb \times Ub$
$(Wa,Wb) \geq 0$

(3) The path angle difference in the compensation plane is large.
(Ra,Rb) < 0

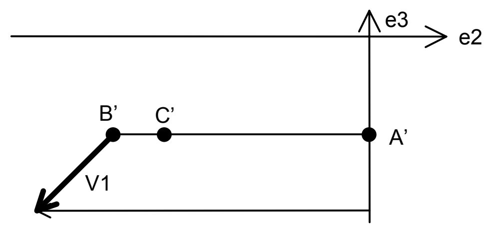
<2> Suppressing the issue of the alarm with a Q command
By inserting a Q command into a block that resulted in the alarm, the issue of the alarm can be suppressed.
(1) Q1 command
By inserting a Q1 command, a perpendicular vector is generated.
Example: N4 Y-200 Z-200 Q1



At B', a vector (V) perpendicular to A'B' is generated.

Fig. 21.4.1.1 (o) Q1 command

A perpendicular vector can also be generated by specifying G41.2 or G42.1 in the next block as follows:
Example: N6 G41.2 Y-400 Z0
(2) Q2 command
With a program specifying a linear-to-linear connection, up to two compensation vectors are generated. In this case, the second vector is deleted by inserting a Q2 command.
The Q2 command has no effect on circular interpolation.
Example: N4 Y-200 Z-200 Q2



The second vector (V2) is deleted, and only V1 is used as a compensation vector.

Fig. 21.4.1.1 (p) Q2 command

(3) Q3 command
By inserting a Q3 command, the issue of the alarm can be suppressed.
Example: N4 Y-200 Z-200 Q3

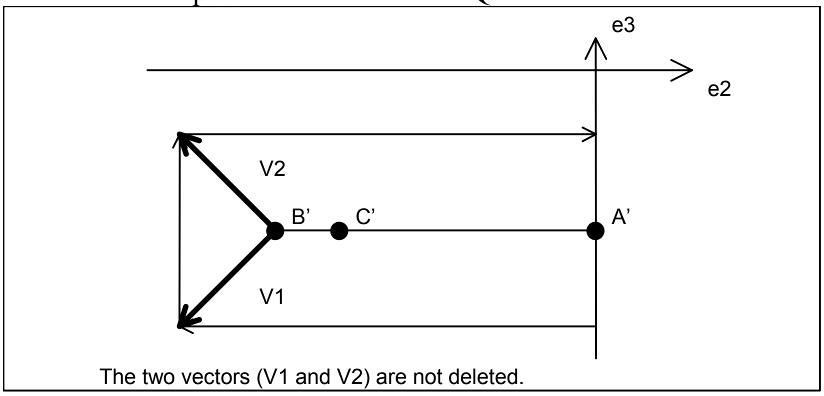


Fig. 21.4.1.1 (q) Q3 command

- Others

When the tool movement changes linear to circular (helical), circular (helical) to linear, or circular (helical) to circular (helical), the start, end, and center points of a circular (helical) movement are projected on the compensation plane that is perpendicular to the tool axis, and a compensation vector is calculated in the plane. The obtained vector is added to the originally specified position to create a position to be specified. Then, the tool is moved linearly or circularly (helically) to the created position.

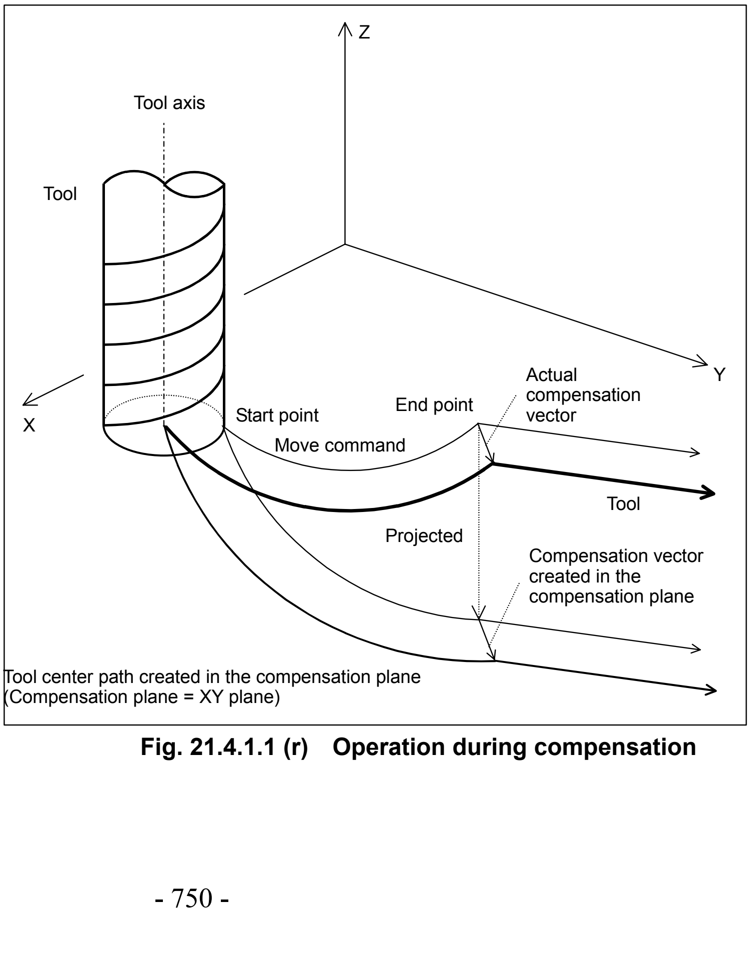


Fig. 21.4.1.1 (r) Operation during compensation

- Angle of the rotary axis for type 2 (when the movement range is not specified)

When the direction of the tool is specified by I, J, K, Q for type 2, more than two pairs of "computed angles" of the rotary axes usually exist.

The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the specified tool axis direction.

The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.

The following descriptions assume that there is no movement range specification (parameter No.19741 - No.19744 = 0).

Tool rotation type or table rotation type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller.
↓ ↓ When the master axis moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller.
↓ ↓ When the slave axis moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
Mixed-type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller.
↓ ↓ When the table moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.
↓ ↓ When the tool moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
↓ ↓ When the table angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Fig. 21.4.1.1 (s) Output judgment conditions

The process of judging whether the moving angle is smaller or larger as the output judgement condition is called "movement judgement."
When parameter PRI (No.19608#5) is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

The "movement judgement" process is explained below.
When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."
Usually, two pairs of "basic computed angles" exist.
For example, assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A $\theta 1$ degree; B $\phi 1$ degree)
(A $\theta 2$ degrees; B $\phi 2$ degrees) where $\theta 1 \leq \theta 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.
Based on the PA angle, the "movement judgement" process is done as follows (when parameter PRI (No.19608#5) is 0).

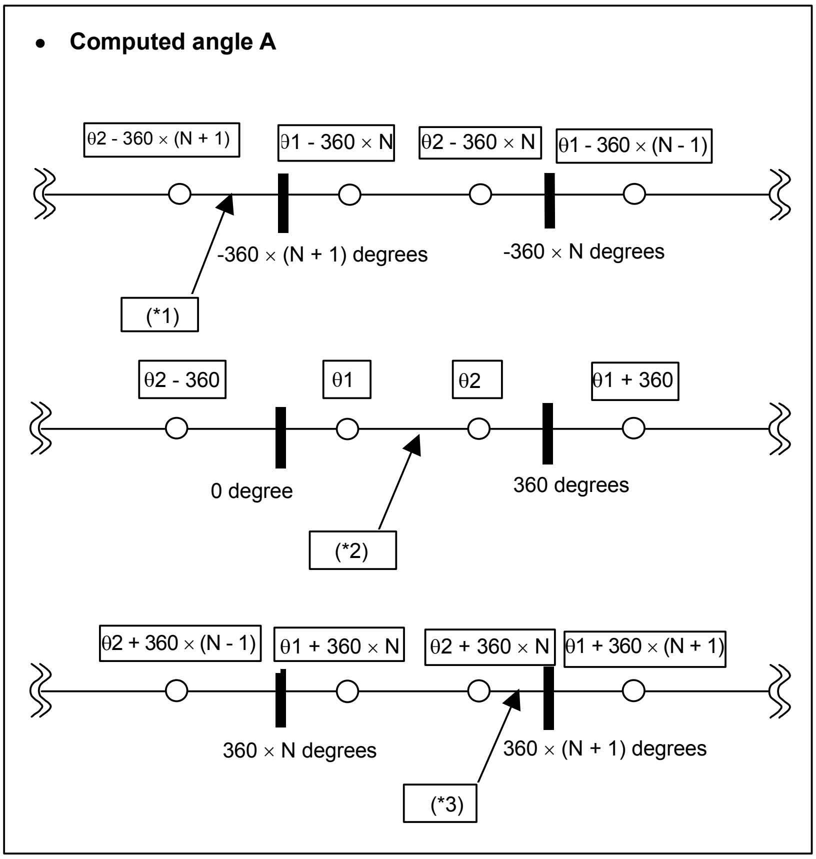


Fig. 21.4.1.1 (t) Movement judgment

When the PA angle is (*1):
The output angle is: (A θ2 - 360 × (N + 1) degrees; B ϕ2 degrees).
Namely, θ2 - 360 × (N + 1) degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2, is adopted as the computed angle of B.

When the PA angle is (*2):
The output angle is: (A θ1 degrees; B ϕ1 degrees).
Namely, θ1 degrees is adopted that is nearer to the computed angle of A, and ϕ1, which is the same group as θ1, is adopted as the computed angle of B.

When the PA angle is (*3):
The output angle is: (A θ2 + 360 × N degrees; B ϕ2 degrees).
Namely, θ2 + 360 × N degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2. is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgement" is made for rotary axis B (slave) according to the "output judgment conditions."
If the "output angle" of rotary axis A is determined by the "movement judgement" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis B.
Similarly, if the "output angle" of rotary axis B is determined by the "movement judgement" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

The "output angle" is explained below using a tool rotation type machine as an example.
This example illustrates a machine having a "BC type tool axis Z."

- **BC type tool axis Z**

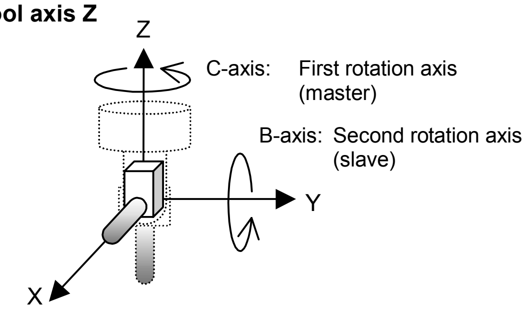


Fig. 21.4.1.1 (u) BC type tool axis Z

The following two pairs of "computed basic angles" exist that direct the tool axis toward the +X-axis direction.
(B 90 degrees; C 180 degrees)
(B 270 degrees; C 0 degree)

<1> When the current rotary axis angles are (B -70 degrees; C 30 degrees)
The "output angles" are (B -90 degrees; C 0 degree).
0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).

<2> When the current rotary axis angles are (B 80 degrees; C 500 degrees)
The "output angles" are (B 90 degrees; C 540 degrees).
540 degrees (180 degrees + 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.

<3> When the current rotary axis angles are (B 60 degrees; C 90 degrees)
The "output angles" are (B 90 degrees; C 180 degrees).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.

<4> When the current rotary axis angles are (B 180 degrees; C 90 degrees)
The "output angles" are (B 270 degrees; C 0 degree).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree.
That is, the pair is adopted whose C-axis angle is 0 degree and whose B-axis angle is 270 degrees.

When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.
In that case, the master axis does not move from the current angle.

An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

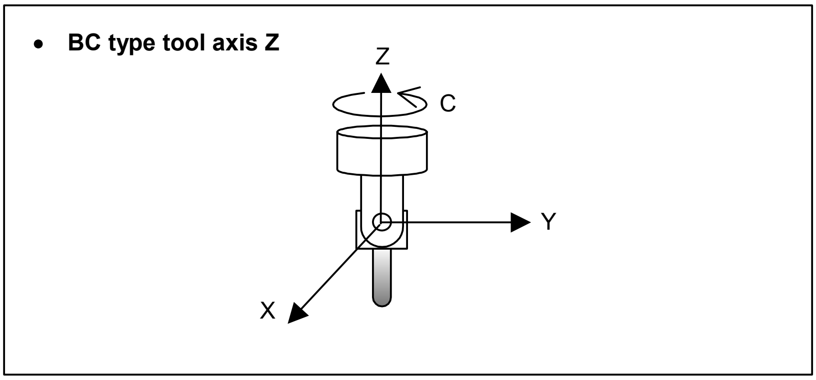


Fig. 21.4.1.1 (v) BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

- Angle of the rotary axis for type 2 (when the movement range is specified)

If the upper and lower limits of the movement range of the rotary axis is specified using parameters No.19741 to No.19744, the rotary axis will move only within the specified range when the direction is specified using I, J, K, Q command for type 2 control.

Although the procedure for determining the angles is the same as that used "when the movement range is not specified," the "output angles" need to be selected from **those computed angles that are within the specified movement range for both axes**.

Tool rotation type or table rotation type machine
<1> Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller represents the "output angles."
↓ ↓ When the master axis moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller.
↓ ↓ When the slave axis moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
Mixed type machine
<1> Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose table (second rotary axis) moving angle is smaller represents the "output angles."
↓ ↓ When the table moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.
↓ ↓ When the tool moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
↓ ↓ When the table angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Fig. 21.4.1.1 (w) Output judgment conditions

When parameter PRI (No.19608#5) is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

> ⚠ **CAUTION**
> 1 If the lower limit of the movement range is larger than the upper limit, alarm PS5459 occurs when G43.5 is specified.
> 2 If no "computed angle" is found within the movement range because the range is too small, alarm PS5459 occurs.
> 3 If 0 is set for both parameters that specify the upper and lower limits of the movement range, the tool operates assuming that there is no range specification.
> 4 When the rotary axis rollover function or rotary axis control function is used (in which case, set parameter No.1260 (amount of movement per rotation of the rotary axis) to 360 degrees), the tool does not move beyond 0 degree (360 degrees) (does not take the shortcut) if the movement range is set between 0 and 360 degrees. Also, do not specify a negative value or a value larger than 360 degrees for the movement range.

An example of the "movement judgement" process is given below.
Assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A $\theta 1$ degree; B $\phi 1$ degree)
(A $\theta 2$ degrees; B $\phi 2$ degrees) where $\theta 1 \le \theta 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
Assume that the current positions and movement ranges of rotary axis A (master) and rotary axis B (slave) are as shown in Fig. 21.4.1.1 (x)、Fig. 21.4.1.1 (y).

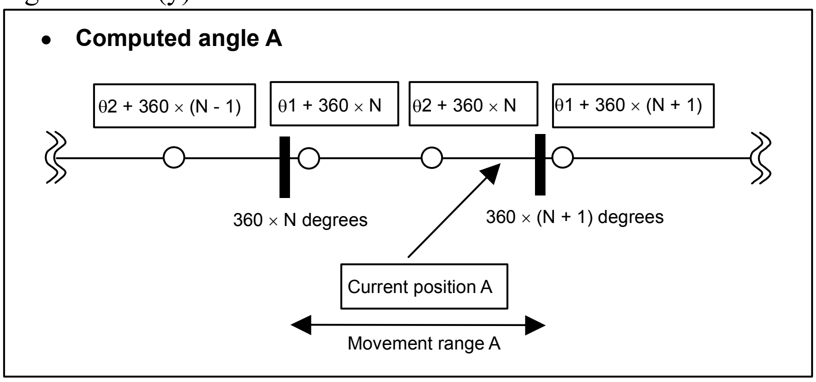


Fig. 21.4.1.1 (x) Computed angle of rotary axis A and its current position and movement range

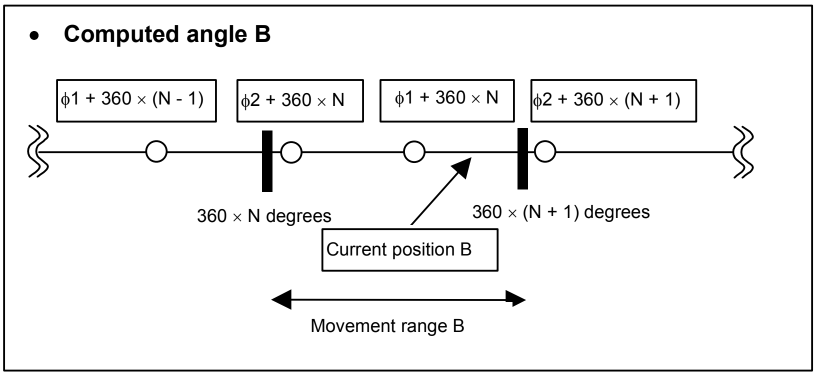


Fig. 21.4.1.1 (y) Computed angle of rotary axis B and its current position and movement range

When the two axes have a positional relationship as shown in the figure, the output angle of rotary axis A is (θ2 + 360 × N) degrees and that of rotary axis B is (ϕ2 + 360 × N) degrees (when parameter PRI (No.19608#5) is set to 0).
More concretely, from the computed angles obtained for rotary axis A, the nearest angle within the movement range, i.e. θ2 + 360 × N degrees, is first adopted. Then, from the computed angles obtained for rotary axis B, the angle belonging to the same group as θ2, i.e. ϕ2 + 360 × N, is adopted.

Note that, in this example, the output angles and moving direction differ depending on whether the movement range is specified or not (0 to 360 degrees), even if N is set to 0 and coordinates are rounded to 0 to 360 degrees.
Namely, if the movement range is not specified, θ1 + 360 degrees nearest to the current position is adopted as the computed angle for rotary axis A and, from the computed angles belonging to the same group as θ1, ϕ1 degrees nearest to the current position is adopted as the computed angle for rotary axis B. Rotary axis A moves in the plus direction. As its coordinate is rounded to 360 degrees, rotary axis A reaches θ1 degrees while moving in the plus direction.
By contrast, when the movement range is set to 0 to 360 degrees, the output angles are (A θ2 degrees; B ϕ2 degrees). Neither rotary axis A nor B moves in a way that it exceeds 0 degree (360 degrees).

21.4.1.2 Leading edge offset

Overview

Leading edge offset is a type of cutter compensation used when a workpiece is machined with the edge of a tool. The tool is automatically shifted by the amount of cutter compensation on the line where a plane formed by a tool vector and tool movement direction meets a plane perpendicular to the tool axis direction.

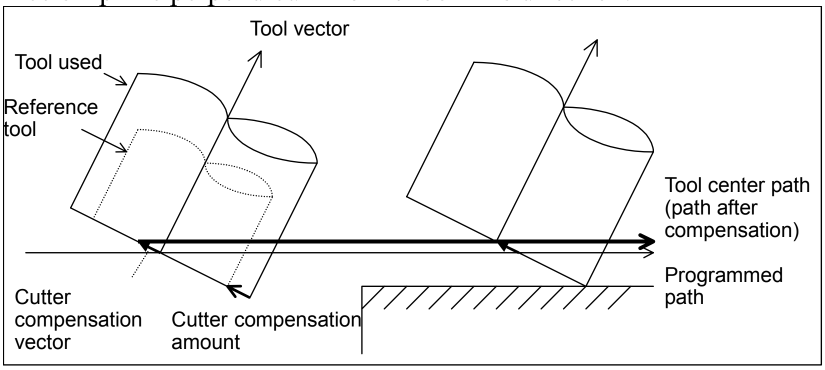


Fig. 21.4.1.2 (a) Leading edge offset

Format

- Leading edge offset

```
G41.3 D_ ;
```

- Canceling the leading edge offset

```
G40 ;
```

NOTE

1 G41.3 can be specified only in the G00 and G01 modes. In a block containing G41.3 or G40, only addresses D, O, and N can be specified.
2 The block that follows a block containing a G41.3 command must contain a move command.
In the block after G41.3, however, a tool movement in the same direction as the tool axis direction or the opposite direction cannot be specified.
3 No modal G code that belongs to the same group as G00 and G01 can be specified in the G41.3 mode. If such a modal G code is specified, alarm PS5460 is issued.
4 Leading edge offset does not provide type 2 commands. It does not allow the tool direction to be specified with the I, J, and K commands.

Explanation

- Operation at startup and cancellation

The operation performed at leading edge offset startup and cancellation does not vary. When G41.3 is specified, the tool is moved by the amount of compensation (Vc) in the plane formed by the movement vector (V_M) of the block after the G41.3 block and the tool vector (V_T) obtained at the time of G41.3 specification. The tool movement is perpendicular to the tool vector. When G40 is specified, the tool is moved to cancel V_C. The following illustrates how the compensation is performed:

<1> When the tool vector is inclined in the tool movement direction

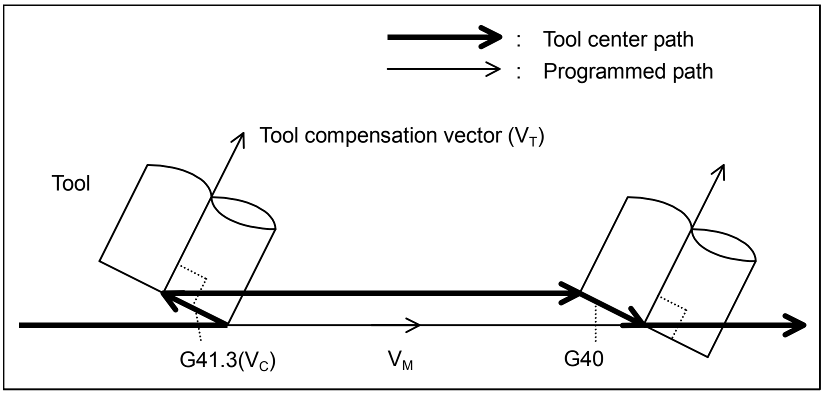


Fig. 21.4.1.2 (b) When the tool vector is inclined in the tool movement direction

<2> When the tool vector is inclined in the direction opposite to the tool movement direction

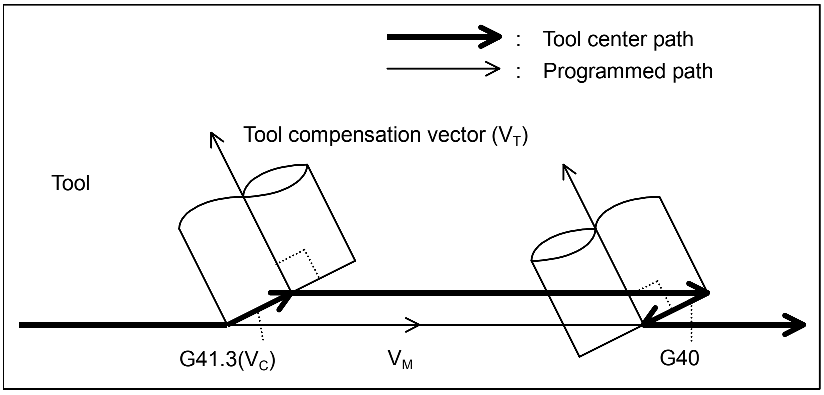


Fig. 21.4.1.2 (c) When the tool vector is inclined in the direction opposite to the tool movement direction

- Operation during compensation

The tool center moves so that a compensation vector (VC) perpendicular to the tool vector (VT) is created in the plane formed by the tool vector (VT) at the end point of each block and the movement vector (VM) of the next block.

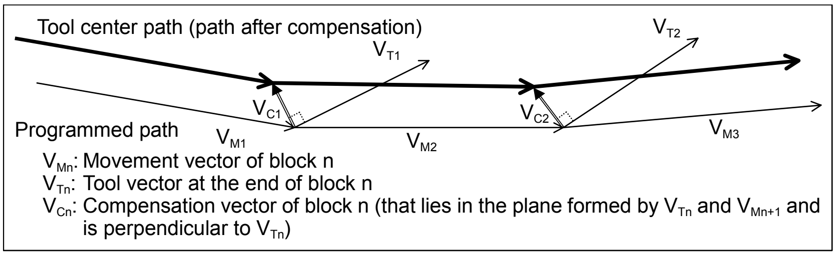


Fig. 21.4.1.2 (d) Operation during compensation

If a G code or M code that suppresses buffering is specified in the compensation mode, however, the compensation vector created immediately before the specification is maintained.

When a block specifying no movement (including a block containing a move command for a rotary axis only) is specified in the compensation mode, the movement vector of the block after the block specifying no movement is used to create a compensation vector as shown below.

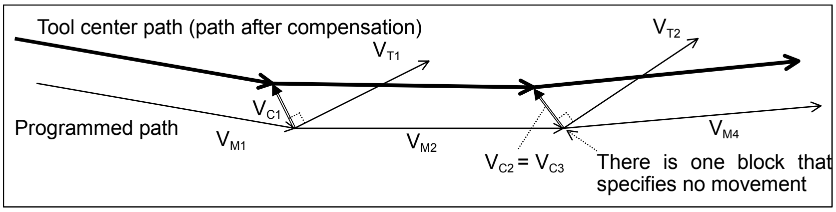


Fig. 21.4.1.2 (e) When there is one block that specifies no movement

If block 3 specifies no movement, the compensation vector of block 2 (VC2) is created in a plane formed by the movement vector of block 4 (VM4) and the tool vector (VT2) at the end of block 2. VC2 is perpendicular to VT2.

> ⚠ **CAUTION**
> If two or more successive blocks specify no movement, the previously created compensation vector is maintained. However, such specification should be avoided.

- Block immediately before the offset cancel command (G40)

In the block immediately before the offset cancel command (G40), a compensation vector is created from the movement vector of that block and the tool vector at the end point of the block as shown below.

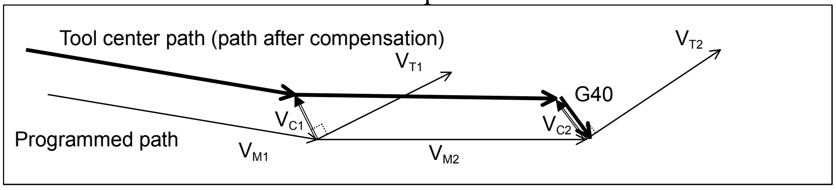


Fig. 21.4.1.2 (f) Block immediately before G40

The compensation vector (VC2) of block 2 is created in a plane formed by the tool vector (VT2) at the end point of block 2 and the movement vector (VM2) of block 2. VC2 is perpendicular to VT2.

- Compensation performed when θ is approximately 0°, 90°, or 180°

When the included angle θ between VMn+1 and VTn is regarded as 0°, 180°, or 90°, the compensation vector is created in a different way. So, when creating a program, note the following points:

<1> Setting a variation range for determining θ to be 0°, 180°, or 90°
When the included angle (θ) between the tool vector (VTn) and movement vector (VMn+1) becomes approximately 0°, 180°, or 90°, the system regards θ as 0°, 180°, or 90°, respectively, then creates a compensation vector which is different from the normal compensation vector. The variation range used for determining θ to be 0°, 180°, and 90° is set in parameter No. 19631. For example, let the angle set in this parameter be Δθ. Then, the system regards θ as follows:

(1) If 0 ≤ θ ≤ Δθ , θ is regarded as 0°.

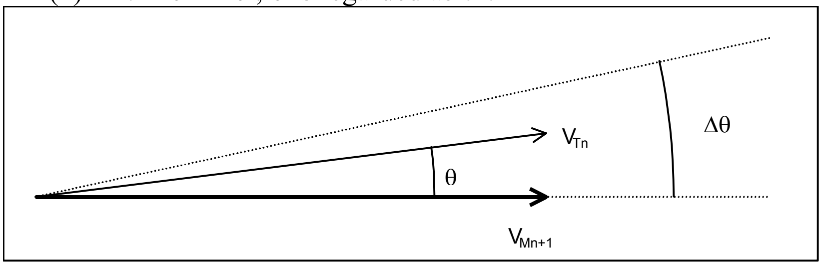


Fig. 21.4.1.2 (g) Determination of θ = 0°

(2) If (180 - Δθ) ≤ θ ≤ 180, θ is regarded as 180°.

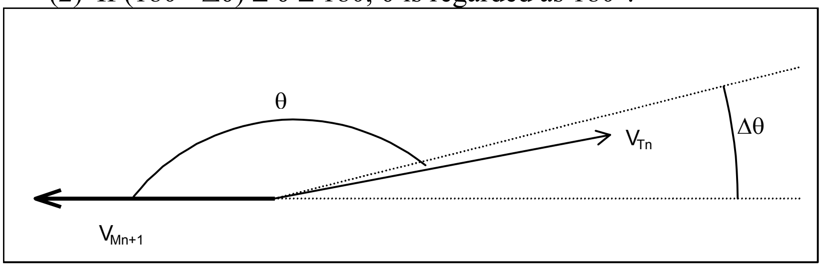


Fig. 21.4.1.2 (h) Determination of θ = 180°

(3) If (90 – Δθ) ≤ θ ≤ (90 + Δθ), θ is regarded as 90°.

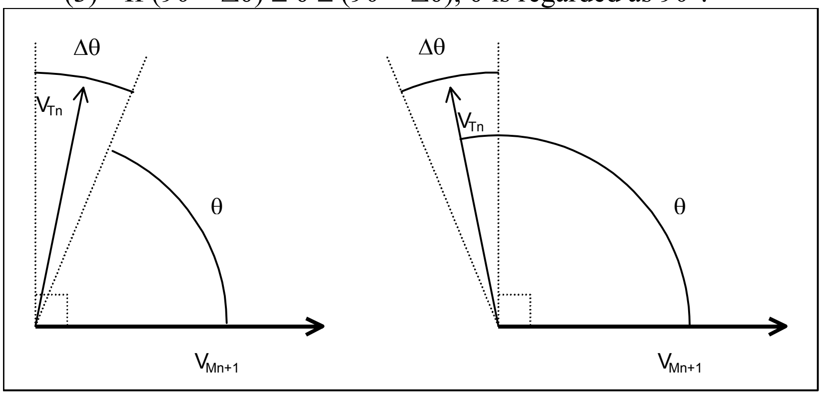


Fig. 21.4.1.2 (i) Determination of θ = 90°

<2> Compensation vector when θ is regarded as 0° or 180°

At startup (when G41.3 is specified), alarm PS5408 is issued.

This means that the tool vector of a block and the movement vector of the next block must not point in the same direction or in opposite directions at startup.

At other than startup, the previously created compensation vector is maintained without change.

If the included angles between VT2 and VM3, VT3 and VM4, and VT4 and VM5 are regarded as 0°, compensation vector VC1 of block 1 is maintained as compensation vectors VC2, VC3, and VC4 of blocks 2, 3, and 4, respectively.

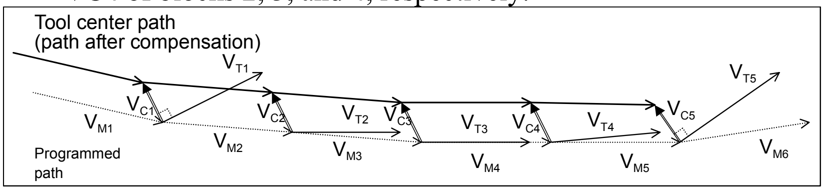


Fig. 21.4.1.2 (j) When θ = 0° is determined

If the included angles between VT2 and VM3, VT3 and VM4, and VT4 and VM5 are regarded as 180°, compensation vector VC1 of block 1 is maintained as compensation vectors VC2, VC3, and VC4 of blocks 2, 3, and 4, respectively.

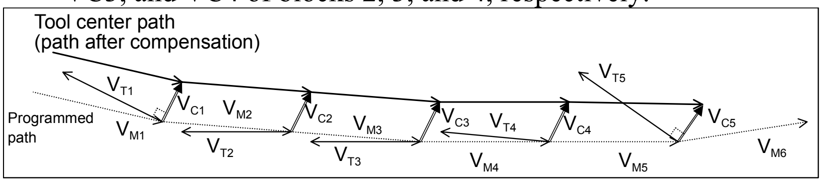


Fig. 21.4.1.2 (k) When θ = 180° is determined

<3> Compensation vector when θ is regarded as 90°

If the previous compensation vector (VCn-1) points in the opposite direction ((VMn × VTn-1) × VTn-1 direction) to VMn with respect to VTn-1, the current compensation vector (VCn) is created so that it also points in the (VMn+1 × VTn) × VTn direction.

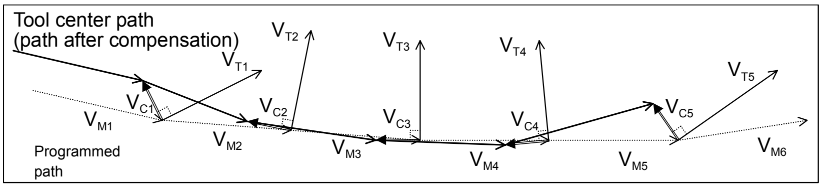


Fig. 21.4.1.2 (l) When θ = 90° is determined 1

If the previous compensation vector (VCn-1) points in the same direction (-(VMn × VTn-1) × VTn-1 direction) as VMn with respect to VTn-1, the current compensation vector (VCn) is created so that it also points in the -(VMn+1 × VTn) × VTn direction.

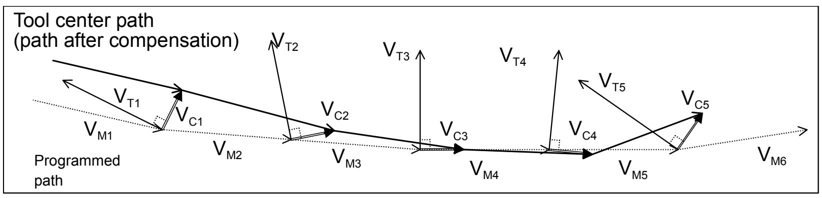


Fig. 21.4.1.2 (m) When θ = 90° is determined 2

21.4.1.3 Tool tip position (cutting point) command

Overview

For machines having a rotary axis for rotating a tool, this function performs cutter compensation for 5-axis machining at the tool tip position if a programmed point is specified with a pivot point.
When this function is used, the programmed point (pivot point) is converted into a tool tip position (cutting point) and a vector of cutter compensation for 5-axis machining is calculated for the position obtained by the conversion. Then, the programmed point (pivot point) is compensated for with the vector of cutter compensation for 5-axis machining.
If the tool side offset (G41.2/G42.2) of cutter compensation for 5-axis machining is performed, the operation of this function is as follows:

(1) If parameter No. 19632 is 0
The vector of cutter compensation for 5-axis machining is calculated at the programmed point (pivot point).
(2) If parameter No. 19632 is not 0 (this function)
The vector of cutter compensation for 5-axis machining is calculated at the tool tip position (cutting point).

Explanation

- Operation explanation

This function calculates a vector at the tool tip position for the cutter compensation function for 5-axis machining as described below.

(1) Convert the programmed coordinates from a programmed point (pivot point) to a tool tip position (cutting point). Parameter No. 19632 is used to store the distance from the programmed point (pivot point) to the tool tip position (cutting point).
(2) Calculate a vector of cutter compensation for 5-axis machining at the tool tip position (cutting point).
(3) Add the cutter compensation vector to the programmed point (pivot point).

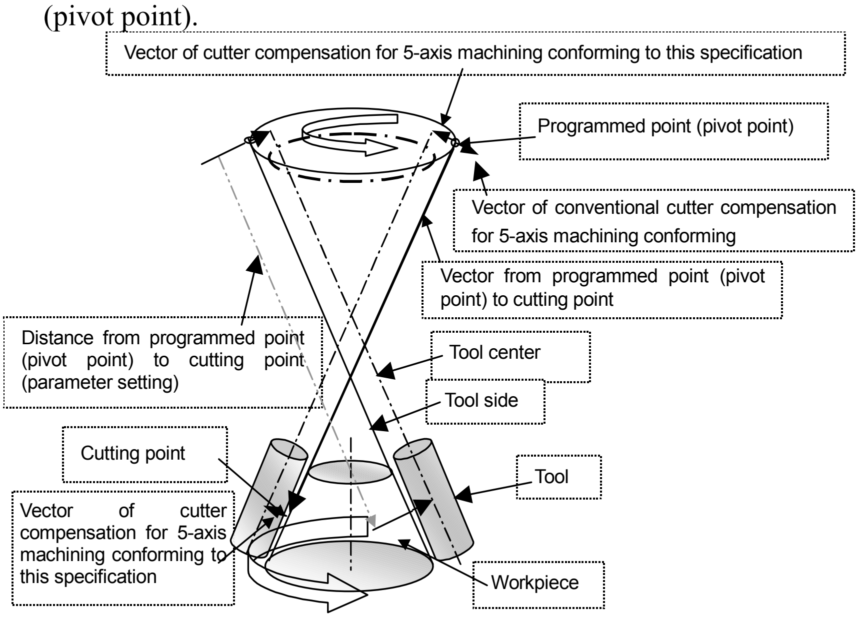


Fig. 21.4.1.3 (a) Basic operation (for G42.2)

- Operation example

For a machine configuration in which the tool axis direction is along the Z-axis and the rotary axes are the B and C axes (Fig. 21.4.1.3 (b))

LC: Parameter (No. 19632) specifying the distance from the programmed point (pivot point) to the tool tip position (cutting point)

b: Specified B-axis value, c: Specified C-axis value

Q = (Qx,Qy,Qz): Programmed point (pivot point)

P, R: Programmed points (pivot points) in the preceding and succeeding blocks

QT = (QTx,QTy,QTz): Tool position (tool tip position (cutting point)) resulting from conversion

PT, RT: Tool positions (tool tip positions (cutting positions)) in the preceding and succeeding blocks

Then,

<1> Convert programmed points (pivot points) P, Q, and R to tool tip positions (cutting points) PT, QT, and RT.

$$QTx = LC \times \sin(b) \times \cos(c) + Qx$$
$$QTy = LC \times \sin(b) \times \sin(c) + Qy$$
$$QTz \quad LC \times \cos(b) + Qz$$

(The same applies to PT and RT.)

<2> Calculate vector VD of cutter compensation for 5-axis machining from tool tip positions (cutting points) PT, QT, and RT and tool gradient VT.

<3> Add cutter vector VD to programmed point (pivot point) Q and set the result as the end point position.

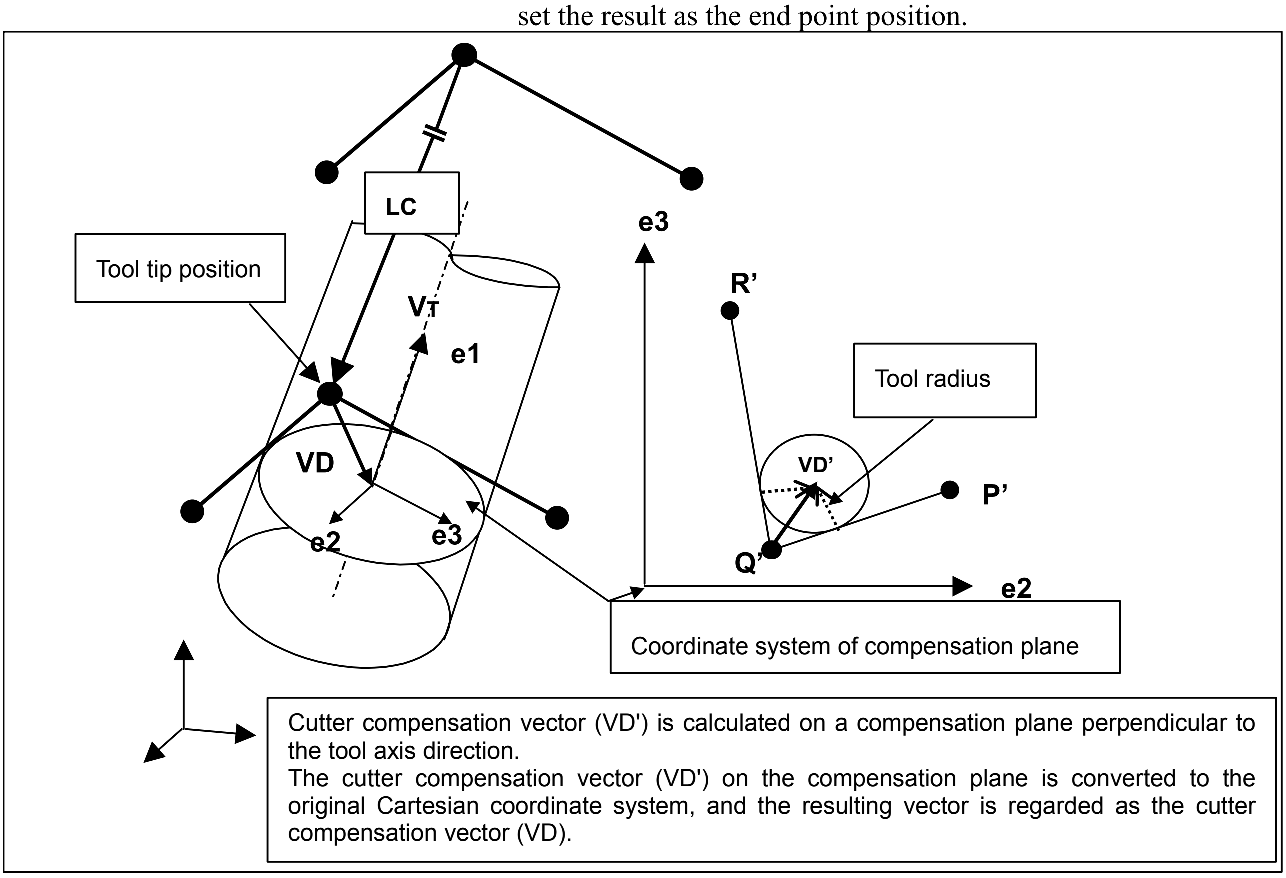


Fig. 21.4.1.3 (b) Operation

⚠ **CAUTION**

1. This function is disabled for leading edge offset.
2. With a command for a rotary axis only, this function does not calculate a cutter compensation vector.
3. This function cannot be used in the three-dimensional coordinate conversion mode.
4. In addition to the cautions given here, the cautions on the cutter compensation function for 5-axis machining apply to this function.

21.4.2 Cutter Compensation in Table Rotation Type Machine

Overview

Cutter compensation can be performed for a 5-axis machine having a rotary table as shown in Fig. 21.4.2 (a).
Shown below is a 5-axis machine that has table rotation axis A on the X-axis and table rotation axis B on the Y-axis.
This machine configuration is used as a sample configuration in the following explanation unless otherwise noted:

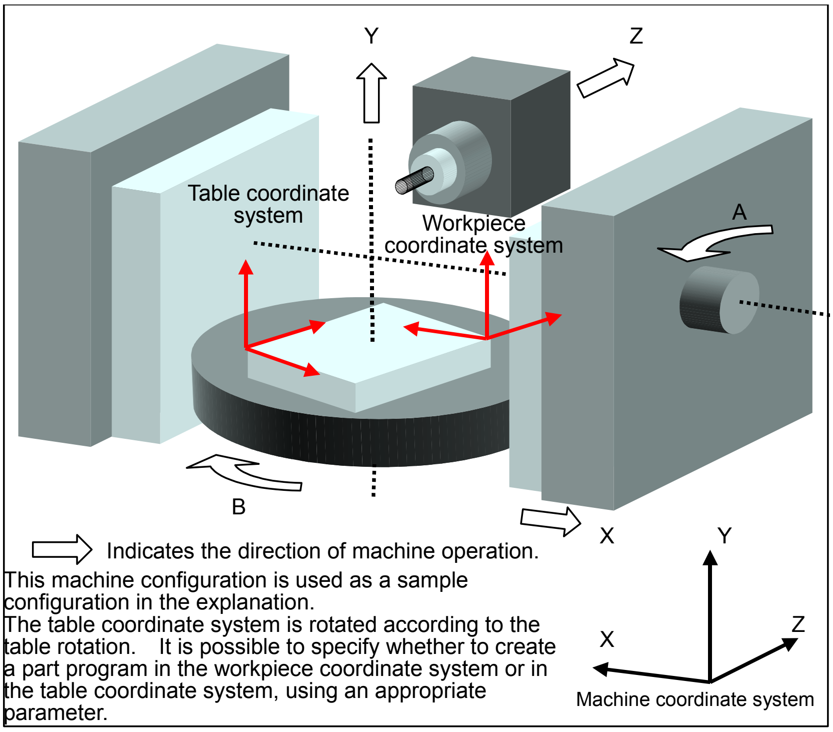


Fig. 21.4.2 (a) Machine having a rotary table

Format

- Startup (start of cutter compensation) (type 1)

When bit 1 (SPG) of parameter No. 19607 is 0

G41.2 (or G42.2) IP_ D_ ;

G41.2:	Cutter compensation left (group 07)
G42.2:	Cutter compensation right (group 07)
IP_:	Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_:	Code specifying the cutter compensation amount (1 to 3 digits)

When bit 1 (SPG) of parameter No. 19607 is 1

G41.4 (or G42.4) IP_ D_ ;

G41.4:	Cutter compensation left (group 07)
G42.4:	Cutter compensation right (group 07)
IP_:	Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_:	Code specifying the cutter compensation amount (1 to 3 digits)

NOTE

1. In a table rotation type machine (parameter No. 19680 = 12), if an attempt is made to issue G41.4 or G42.4 with SPG, bit 1 of parameter No. 19607, equal to 0, alarm PS0010 is generated.
2. In a table rotation type machine, if an attempt is made to issue G41.2 or G42.2 with SPG, bit 1 of parameter No. 19607, equal to 1, alarm PS5460 is generated.
3. In machine not of the table rotation type machine, if an attempt is made to issue G41.4 or G42.4 with SPG, bit 1 of parameter No. 19607, equal to 1, alarm PS5460 is generated.

- Startup (start of cutter compensation) (type 2)

G41.6(or G42.6) IP_ D_ Q_ ;
IP_ I_ J_ K_ ;
:

G41.6: Cutter compensation left (group 07)
G42.6: Cutter compensation right (group 07)
IP_: Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_: Code specifying the cutter compensation amount (1 to 3 digits)
Q_: Tool's angle of gradient (in degrees)
I_ J_ K_: Tool axis direction at the block end point as viewed from the programming coordinate system

For type 2, do not specify a rotation axis but specify the direction at the tool end point as viewed from the programming coordinate system (workpiece coordinate system), with I, J, and K. Specifying a rotation axis causes alarm PS5460 to be generated.

In a tool rotation type machine, I, J, and K can be specified in a G41.6/G42.6 command block; in a table rotation type machine, however, they cannot. If an attempt is made to specify them, alarm PS5460 is generated.
The following are the notes on type 2.

NOTE

1. If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
2. In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
3. If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
4. If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. 1260 (mount of travel per rotation about the rotation axis).
5. They can be used only with the settings that select the table coordinate system as a programming coordinate system (WKP, bit 5 of parameter No. 19696, = 0 and TBP, bit 4 of parameter No. 19746, = 1). If an attempt is made to issue G41.6/G42.6 with the settings that select the workpiece coordinate system as a programming coordinate system, alarm PS5460 is generated.

- Canceling the cutter compensation

G40 IP_ ;

G40: Cutter compensation cancellation (group 07)

IP_: Value specified for axis movement

- Selecting an offset plane

When parameter PTC (No. 19746) is 1, compensation is performed on the selected plane, on the assumption that the tool is pointing to the direction perpendicular to that plane.

Offset plane	Plane selection command	IP_
XpYp	G17 ;	Xp_Yp_
ZpXp	G18 ;	Xp_Zp_
YpZp	G19 ;	Yp_Zp_

(Example: In the case of Fig. 21.4.2 (a), the XpYp plane is selected.)
The two axes of a selected plane must be included in the three basic axes (the axes for which parameter No. 1022 is set to 1 to 3).

When parameter PTC (No. 19746) is 0, compensation is performed on the plane perpendicular to the tool direction specified with parameters Nos. 19697, 19698, and 19699, regardless of which plane is selected.

NOTE

This function is enabled for tool side offset only. If leading edge offset is specified, alarm PS5460 is generated.

Explanation

- Tool's angle of gradient in type 2

For type 2 of cutter compensation for 5-axis machining, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Fig. 21.4.2 (b).)
Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command.
If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary.

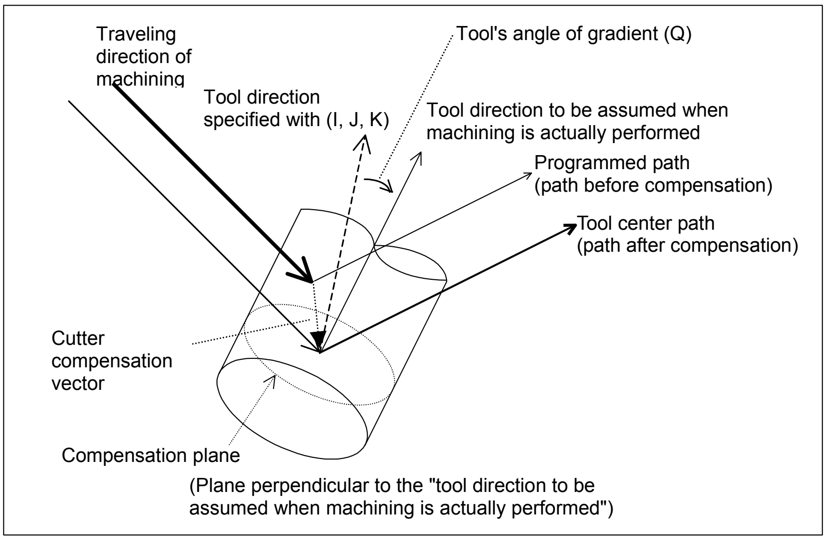


Fig. 21.4.2 (b) Tool's angle of gradient in type 2

- Cutter compensation

The cutter compensation function in table rotation type machines basically performs operations in conformance with cutter compensation. The operations different from those of cutter compensation are mainly described below. For the specifications and cautions not mentioned here, see the description of cutter compensation.

- Startup

When cutter compensation for the rotary table is specified (G41.2 or G42.2, G41.4 or G42.4, a dimension word other than 0 in the offset plane, or a D code other than D0) in the offset cancel mode, the CNC enters the offset mode.
Startup is specified with positioning (G00) or linear interpolation (G01).

> **NOTE**
> If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified at startup, alarm PS0034 is issued.

- Commands in the offset mode

In the offset mode, compensation is performed for positioning (G00) and linear interpolation (G01).

> **NOTE**
> If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified in the mode for cutter compensation for 5-axis machining in a table rotation type machine, alarm PS5460 is issued.

- Offset mode cancellation

If a block satisfying either of the following conditions is executed in the offset mode, the CNC enters the offset cancel mode:

1. G40 is specified.
2. 0 is specified as a code specifying the cutter compensation amount (D code).

When offset cancellation is to be performed, neither circular (G02 or G03) nor involute (G02.2 or G03.2) command can be specified. If offset cancellation is specified in such a mode, alarm PS0034 is issued.

- If selecting the table coordinate system as a programming coordinate system

If TBP, bit 4 of parameter No. 19746, is 1 and WKP, bit 5 of parameter No. 19696, is 0, specifying cutter compensation for 5-axis machining causes the table coordinate system to be selected as a programming coordinate system. The table coordinate system refers to the workpiece coordinate system fixed to the table when cutter compensation for 5-axis machining.

In the blocks subsequent to the one in which cutter compensation for 5-axis machining is specified, the table coordinate system rotates with the rotation of the table.

A linear axis (X, Y, Z) command is assumed to be issued on the table coordinate system.

Specifying linear interpolation causes cutter compensation to be performed on the linear interpolation command on the table coordinate system.

The table coordinate system does not rotate with the rotation of the tool head.

The cancel (G40) block becomes a block that suppresses buffering.

It is possible to switch between absolute and relative coordinate displays, using DET, bit 2 of parameter No. 19608, as follows:

- When DET is 0, the position in the table coordinate system is displayed.
- When DET is 1, the position in the workpiece coordinate system is displayed.

Note, however, the distance to go is always that in the programming coordinate system.

NOTE

1 Either the AI contour control I or AI contour II control option is required. In addition, be sure to specify the following parameters:
 (1) LRP, bit 1 of parameter No. 1401, = 1: Linear rapid traverse
 (2) FRP, bit 5 of parameter No. 19501, = 1: Acceleration/deceleration before interpolation is used for rapid traverse.
 (3) Parameter No. 1671: Acceleration of acceleration/deceleration before interpolation for rapid traverse
 (4) Parameter No. 1672: Change time for bell-shaped acceleration/deceleration before interpolation for rapid traverse
 (5) Parameter No. 1660: Maximum permissible acceleration for acceleration/deceleration before interpolation

 If they are not specified, alarm PS5483 is generated.

NOTE

2 When table rotation axis movement is specified in the start block of cutter compensation for 5-axis machining, after the movement is completed, the workpiece coordinate system is fixed to the table and assumed to be a table coordinate system.

3 In the mode of cutter compensation for 5-axis machining, do not change the workpiece coordinate system or change the workpiece offset value. If an attempt is made to specify workpiece coordinate system selection (G54 to G59), alarm PS5460 is generated.

- If selecting the workpiece coordinate system as a programming coordinate system

If TBP, bit 4 of parameter No. 19746, is 0 or if TBP, bit of parameter No. 19746, is 1 and WKP, bit 5 of parameter No. 19696, is 1, the programming coordinate system does not rotate with the rotation of the table, being fixed to the workpiece coordinate system.

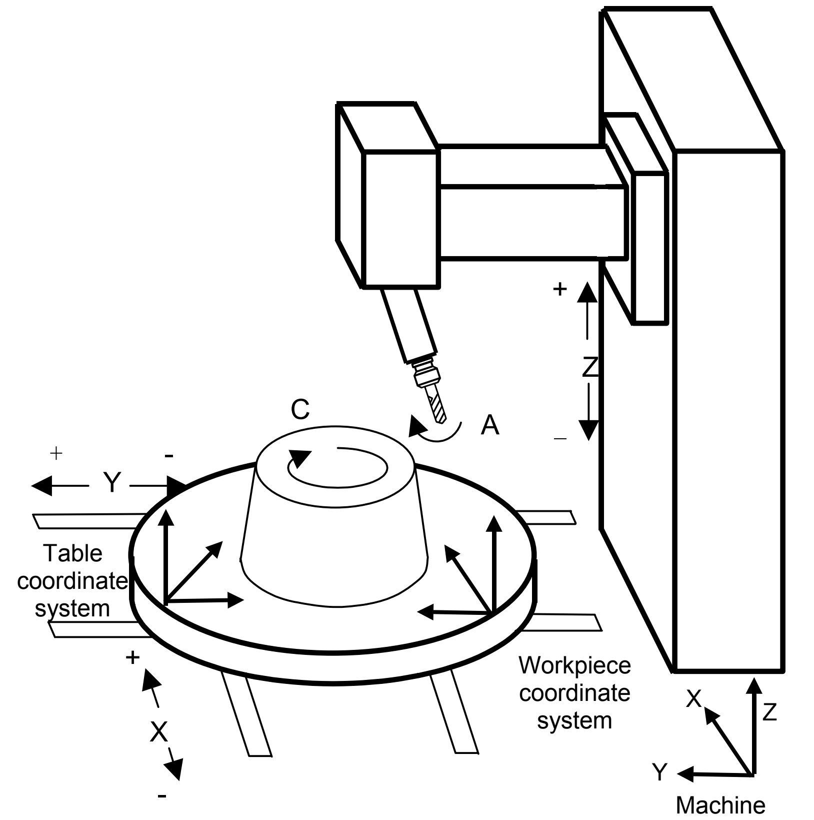
- Angle of the rotation axis in type 2

For an explanation of how the rotation axis end point is determined when the tool direction is specified with an IJKQ command in type 2, see the explanations of cutter compensation in a tool rotation type machine, "Angle of the rotation axis in type 2 (if the operating range is not specified)" and "Angle of the rotation axis in type 2 (if the operating range is specified)".

21.4.3 Cutter Compensation in Mixed-Type Machine

Overview

This function can perform three-dimensional cutter compensation in a 5-axis machine having a rotary table and a tool axis as shown in Fig. 21.4.3 (a).
Shown below is a 5-axis machine that has tool axis A on the X-axis (the tool axis direction is along the Z-axis) and table rotation axis B on the Y-axis.
This machine configuration is used as a sample configuration in the following explanation unless otherwise noted:



The table coordinate system is rotated according to the table rotation. It is possible to specify whether to create a part program in the workpiece coordinate system or in the table coordinate system, using an appropriate parameter.

Fig. 21.4.3 (a) Machine having a tool rotation axis and table rotation axis

Format

- Startup (start of cutter compensation (for mixed-type machine configuration)) (type 1)

When bit 1 (SPG) of parameter No. 19607 is 0

G41.2 (or G42.2) IP_ D_ ;

G41.2: Cutter compensation left (group 07)
G42.2: Cutter compensation right (group 07)
IP_: Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_: Code specifying the cutter compensation amount (1 to 3 digits)

When bit 1 (SPG) of parameter No. 19607 is 1

G41.5 (or G42.5) IP_ D_ ;

G41.5: Cutter compensation left (group 07)
G42.5: Cutter compensation right (group 07)
IP_: Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_: Code specifying the cutter compensation amount (1 to 3 digits)

NOTE

1 In a mixed-type machine (parameter No. 19680 = 21), if an attempt is made to issue G41.5 or G42.5 with SPG, bit 1 of parameter No. 19607, equal to 0, alarm PS0010 is generated.
2 In a mixed-type machine, if an attempt is made to issue G41.2 or G42.2 with SPG, bit 1 of parameter No. 19607, equal to 1, alarm PS5460 is generated.
3 In machine not of the mixed-type machine, if an attempt is made to issue G41.5 or G42.5 with SPG, bit 1 of parameter No. 19607, equal to 1, alarm PS5460 is generated.

- Startup (start of cutter compensation (for mixed-type machine configuration)) (type 2)

G41.6 (or G42.6) IP_ D_ Q_ ;
IP_ I_ J_ K_ ;
:

G41.6: Cutter compensation left (group 07)
G42.6: Cutter compensation right (group 07)
IP_: Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_: Code specifying the cutter compensation amount (1 to 3 digits)
Q_: Tool's angle of gradient (in degrees)
I_ J_ K_: Tool axis direction at the block end point as viewed from the programming coordinate system

In a mixed-type machine, I, J, and K can be specified in a G41.6/G42.6 command block; in a table rotation type machine, however, they cannot. If an attempt is made to specify them, alarm PS5460 is generated.
The following are the notes on type 2.

> **NOTE**
> 1 If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
> 2 In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
> 3 If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
> 4 If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. 1260 (mount of travel per rotation about the rotation axis).
> 5 They can be used only with the settings that select the table coordinate system as a programming coordinate system (WKP, bit 5 of parameter No. 19696, = 0 and TBP, bit 4 of parameter No. 19746, = 1). If an attempt is made to issue G41.6/G42.6 with the settings that select the workpiece coordinate system as a programming coordinate system, alarm PS5460 is generated.

- Canceling the cutter compensation

G40 IP_ ;
G40: Cutter compensation cancellation (group 07)
IP_: Value specified for axis movement

> **NOTE**
> This function is enabled for tool side offset only. If leading edge offset is specified, alarm PS5460 is generated.

Explanation

- Tool's angle of gradient in type 2

For type 2 of cutter compensation for 5-axis machining, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Fig. 21.4.2 (b).)
Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command.
If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary.

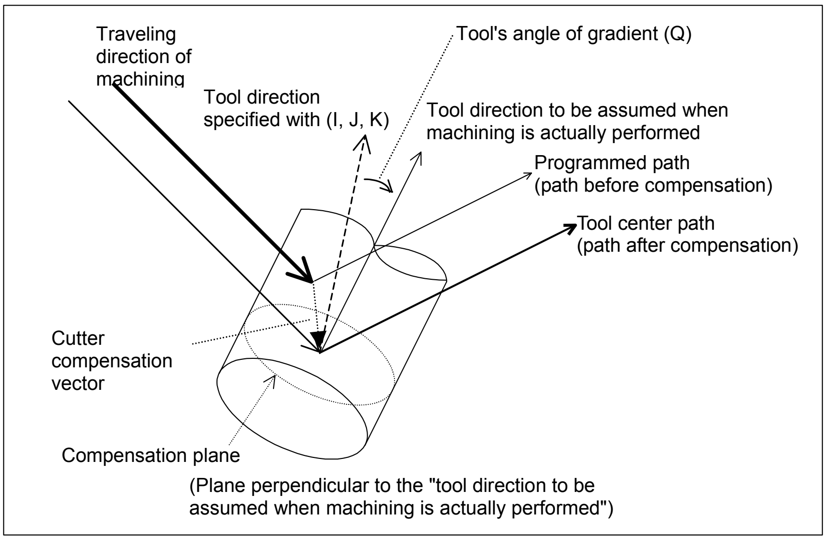


Fig. 21.4.3 (b) Tool's angle of gradient in type 2

- Cutter compensation

The function for cutter compensation for 5-axis machining in a mixed type machine basically performs operations in conformance with cutter compensation for 5-axis machining in a tool rotation type machine. The operations different from those of cutter compensation for 5-axis machining in a tool rotation type machine are mainly described below.
For the specifications and cautions not mentioned here, see the description of cutter compensation for 5-axis machining in a tool rotation type machine.

- Startup

When cutter compensation for 5-axis machining in a mixed type machine (G41.2 or G42.2, G41.5 or G42.5, or a D code other than D0) is specified in the offset cancel mode, the CNC enters the offset mode.

Startup is specified with positioning (G00) or linear interpolation (G01).

> **NOTE**
> If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified at startup, alarm PS0034 is issued.

- Commands in the offset mode

In the offset mode, compensation is performed for positioning (G00) and linear interpolation (G01).

> **NOTE**
> If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified in the mode for cutter compensation for 5-axis machining in a mixed type machine, alarm PS5460 is issued.

- Offset mode cancellation

If a block satisfying either of the following conditions is executed in the offset mode, the CNC enters the offset cancel mode:

1 G40 is specified.

2 0 is specified as a code specifying the cutter compensation amount (D code).

When offset cancellation is to be performed, neither circular (G02 or G03) nor involute (G02.2 or G03.2) command can be specified. If offset cancellation is specified in such a mode, alarm PS0034 is issued.

> **NOTE**
> 1 This function cannot be used in the three-dimensional coordinate conversion mode.
> 2 In addition to the cautions given here, the cautions on the cutter compensation function for 5-axis machining apply to this function.

- If selecting the table coordinate system as a programming coordinate system

If TBP, bit 4 of parameter No. 19746, is 1 and WKP, bit 5 of parameter No. 19696, is 0, specifying cutter compensation for 5-axis machining causes the table coordinate system to be selected as a programming coordinate system. The table coordinate system refers to the workpiece coordinate system fixed to the table when cutter compensation for 5-axis machining.
In the blocks subsequent to the one in which cutter compensation for 5-axis machining is specified, the table coordinate system rotates with the rotation of the table.
A linear axis (X, Y, Z) command is assumed to be issued on the table coordinate system.
Specifying linear interpolation causes cutter compensation to be performed on the linear interpolation command on the table coordinate system.
The table coordinate system does not rotate with the rotation of the tool head.

The cancel (G40) block becomes a block that suppresses buffering.

It is possible to switch between absolute and relative coordinate displays, using DET, bit 2 of parameter No. 19608, as follows:

- When DET is 0, the position in the table coordinate system is displayed.
- When DET is 1, the position in the workpiece coordinate system is displayed.

Note, however, the distance to go is always that in the programming coordinate system.

NOTE

1 Either the AI contour control I or AI contour II control option is required. In addition, be sure to specify the following parameters:
 (1) LRP, bit 1 of parameter No. 1401, = 1: Linear rapid traverse
 (2) FRP, bit 5 of parameter No. 19501, = 1: Acceleration/deceleration before interpolation is used for rapid traverse.
 (3) Parameter No. 1671: Acceleration of acceleration/deceleration before interpolation for rapid traverse
 (4) Parameter No. 1672: Change time for bell-shaped acceleration/deceleration before interpolation for rapid traverse
 (5) Parameter No. 1660: Maximum permissible acceleration for acceleration/deceleration before interpolation
 If they are not specified, alarm PS5483 is generated.

> **NOTE**
> 2 When table rotation axis movement is specified in the start block of cutter compensation for 5-axis machining, after the movement is completed, the workpiece coordinate system is fixed to the table and assumed to be a table coordinate system.
> 3 In the mode of cutter compensation for 5-axis machining, do not change the workpiece coordinate system or change the workpiece offset value. If an attempt is made to specify workpiece coordinate system selection (G54 to G59), alarm PS5460 is generated.

- If selecting the workpiece coordinate system as a programming coordinate system

If TBP, bit 4 of parameter No. 19746, is 0 or if TBP, bit of parameter No. 19746, is 1 and WKP, bit 5 of parameter No. 19696, is 1, the programming coordinate system does not rotate with the rotation of the table, being fixed to the workpiece coordinate system.

- Angle of the rotation axis in type 2

For an explanation of how the rotation axis end point is determined when the tool direction is specified with an IJKQ command in type 2, see the explanations of cutter compensation in a tool rotation type machine, "Angle of the rotation axis in type 2 (if the operating range is not specified)" and "Angle of the rotation axis in type 2 (if the operating range is specified)".

21.4.4 Interference Check and Interference Avoidance

Overview

By setting NI5, bit 1 of parameter No. 19608, to 1, this function performs an interference check on the plane (compensation plane) perpendicular to the tool axis direction regardless of the machine configuration.
If CAV, bit 5 of parameter No. 19607, is set to 1, a vector is generated to avoid interference on the same plane.

Explanation

- For a tool rotation type machine

An interference check is performed, as well as interference avoidance, with the tool path as projected from the workpiece coordinate system (X-Y-Z) onto the compensation plane (X'-Y'-Z') and a compensation vector.

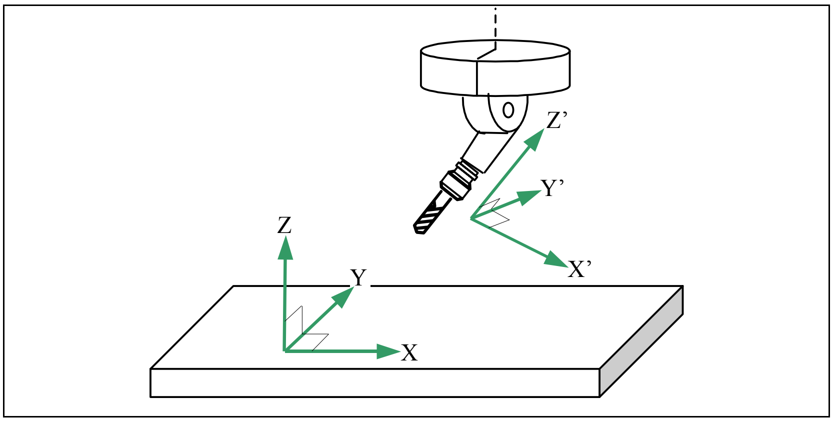


Fig. 21.4.4 (c) Tool rotation type

- For a table rotation type machine

An interference check is performed, as well as interference avoidance, with the tool path as converted from the workpiece coordinate system (X-Y-Z) into the table coordinate system (X'-Y'-Z') and a compensation vector.

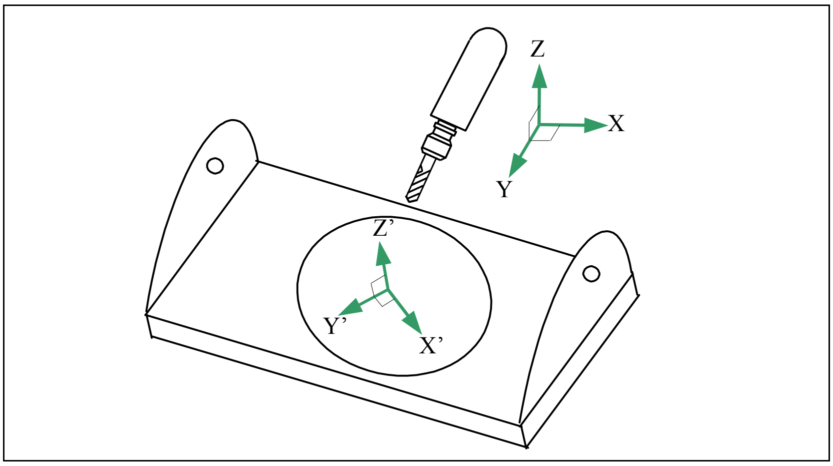


Fig. 21.4.4 (d) Table rotation type

- For a mixed- type machine

An interference check is performed, as well as interference avoidance, with the tool path as projected from the workpiece coordinate system (X-Y-Z) onto the table coordinate system (X'-Y'-Z') and then onto the compensation plane perpendicular to the tool axis direction (X"-Y"-Z") and a compensation vector.

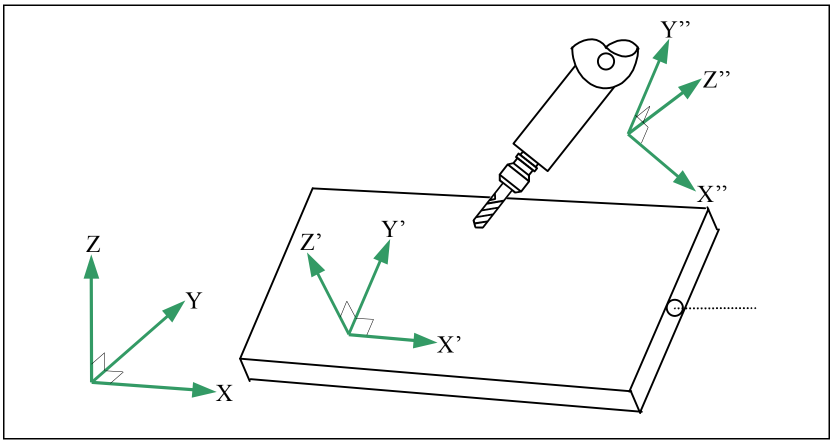


Fig. 21.4.4 (e) Mixed- type

- Interference avoidance

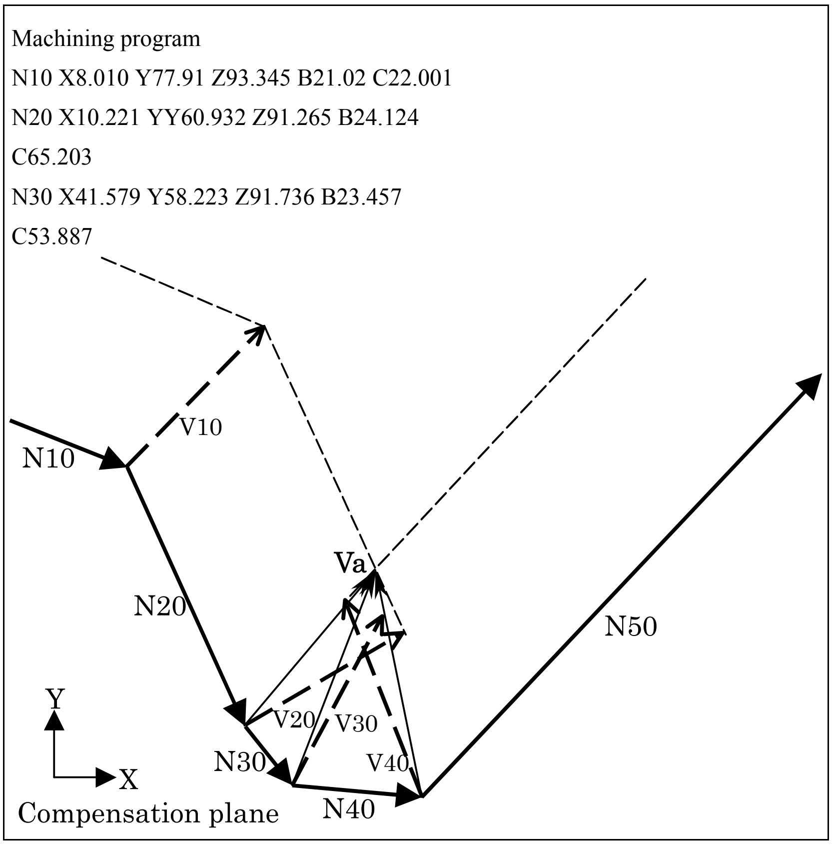


Fig. 21.4.4 (f) Example of interference avoidance

Fig. 21.4.4 (f) shows a tool path in the workpiece coordinate system as projected onto the compensation plane.
For interference avoidance, calculation is performed with the tool path resulting from looking at up to four blocks ahead. At the start of the execution of the N10 block, the system looks at N20 to N50 ahead and generates V20 to V40.
Then, because the movement direction of N30 greatly differs from the direction from V20 to V30, V20 and V30 are considered interferences and eliminated. Similarly, because the movement direction from N30 to N40 greatly differs from the movement direction from V20 to V40, V40 is also considered an interference and eliminated.
Then, the interference avoidance vector Va is generated between N20 and N50 and use it instead of V20, V30, and V40. At this time, because N20 and N50 generally do not intersect, the plane perpendicular to the tool axis direction at the N20 end point is regarded as a compensation plane, N20 and N50 are projected onto this plane, and Va is determined by calculating the intersection point.

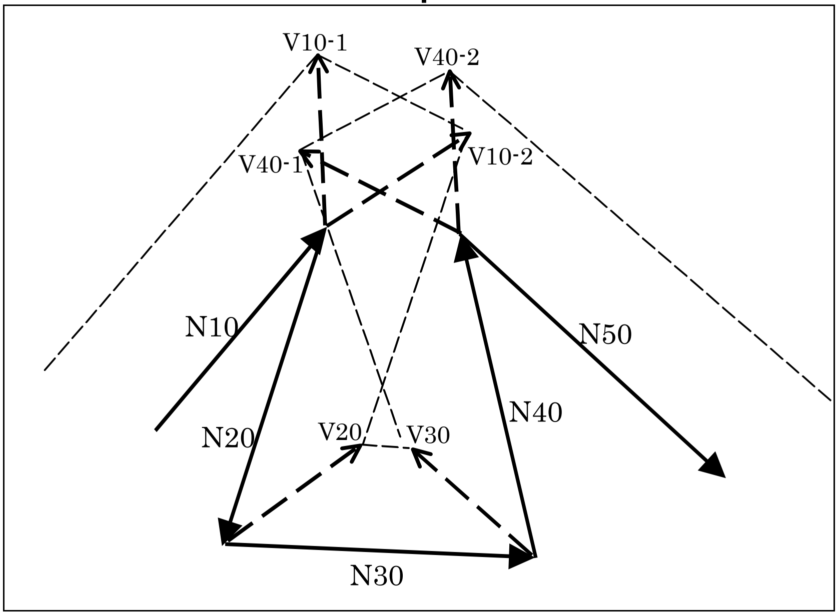
NOTE

Strictly speaking, if the tool axis direction at the N20 end point differs from the tool axis direction at the N50 start point, correct intersection point calculation is not possible. For this reason, the maximum permissible angle by which the tool axis directions in the two blocks used to determine the interference avoidance vector may differ can be set in parameter No. 19636, and if the change in tool axis direction is within the range, an approximate interference avoidance vector is calculated.
If the maximum angle is exceeded, the compensation vector is determined but interference avoidance is not performed.

- If interference avoidance is not possible

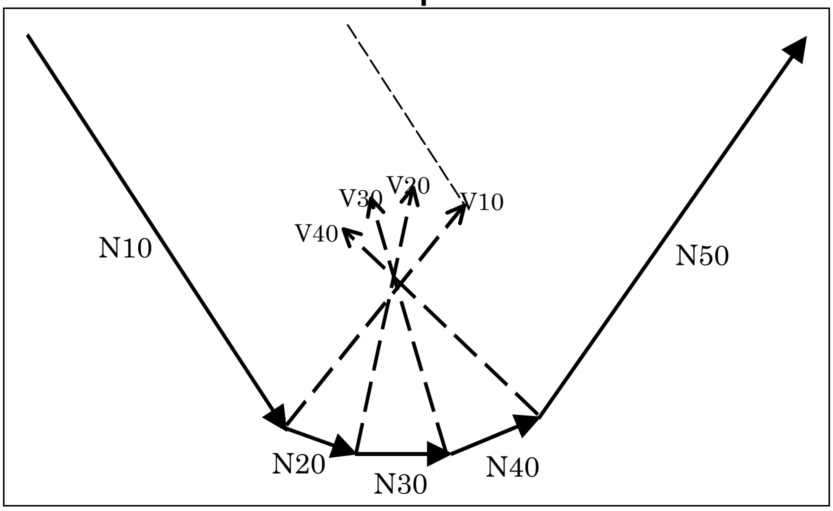
If there are three consecutive interfering blocks, no interference vector can be generated.

Example 1 in which interference avoidance is not possible



N20 to N40 interfere, so that no interference avoidance vector can be generated. Too much cutting results.

Example 2 in which interference avoidance is not possible



N10 to N40 interfere, so that no interference avoidance vector can be generated. V10 causes an interference alarm.

21.4.5 Restrictions

21.4.5.1 Restrictions common to machine configurations

- Interference check

In the mode for cutter compensation for 5-axis machining, interference checks are made using a specified position in the workpiece coordinate system and a compensation vector. The interference check avoidance function cannot be used.

- Corner rounding (G39)

In the mode for cutter compensation for 5-axis machining, G39 cannot be specified. Specifying G39 causes an alarm.

- Reset

Whenever a reset is made in the mode for cutter compensation for 5-axis machining (G41.2, G42.2, G41.4, G42.4, G41.5, or G42.5), the cancel mode (G40) is entered.

- AI contour control I and II

To set the AI contour control I or II mode, the corresponding G code must be specified. Specifying cutter compensation for 5-axis machining does not automatically place the CNC in the AI contour control I or II mode.

- Restricted commands

In the mode for cutter compensation for 5-axis machining, the following functions are available, but their statuses cannot be changed:

- Inch/metric input (If an attempt is made to change the status by using G20 or G21, alarm PS5000 is issued.)
- Mirror image (The signal status cannot be changed.)
- One-digit F code feed (The feedrate cannot be changed by using the manual handle.)

- Unavailable commands

In the mode for cutter compensation for 5-axis machining, the functions listed below cannot be specified. Specifying any of these functions results in an alarm.

- Hypothetical axis interpolation..G07
- Circular interpolation..G07.1
- Polar coordinate interpolation G12.1, G13.1
- Polar coordinate command G15, G16
- Reference position return check G27
- Reference position return G28, G29, G30
- Skip...G31
- Threading...G33
- Automatic tool length measurement.......................................G37
- Normal direction control..............................G40.1, G41.1, G42.1
- Cutter or tool nose radius compensation G41, G42, G39
- Three-dimensional cutter compensation.................................G41
- Wheel wear compensation..G41
- Tool offset...G45, G46, G47, G48
- Programmable mirror image...................................G50.1, G51.1
- Local coordinate system G52
- Machine coordinate system G53
- Workpiece coordinate system setting G54-G59, G54.1
- Rotary table dynamic fixture offset G54.2
- Single direction positioning..G60
- Automatic corner override..G62
- Tapping mode..G63
- Three-dimensional coordinate conversion......................G68, G69
- Tilted working plane command..................................G68.2, G69
- Figure copy...G72.1, G72.2
- Canned cycle.......................G73-G79, G80, G81-G89, G98, G99
- Electric gear box...G80, G81
- Hobbling machine function G80, G81
- External operation function G81
- Chopping G81.1
- Small hole peck drilling cycle G83
- Workpiece coordinate system setting G92
- Workpiece coordinate system presetting G92.1
- Feed per revolution...G95
- Constant surface speed control G96, G97

- Unavailable functions

If the following function is specified in the cutter compensation mode for 5-axis machining, a warning message is issued:

- MDI interruption

If one of the following functions is specified in the cutter compensation mode for 5-axis machining, a PS alarm is issued:

- Manual interruption operation
- Tool retract and recover

In the cutter compensation mode for 5-axis machining, the following functions cannot be used:

- Twin table control
- Flexible synchronous control
- Sequence number comparison and stop (It is not possible to cause a stop by sequence number in the cutter compensation mode for 5-axis machining.)
- Index table indexing
- Retrace function (Programs using the cutter compensation mode for 5-axis machining must not use the retrace function.)
- Rotary axis control
- Manual handle interruption
- External deceleration (External deceleration is not performed.)
- Angular axis control

Combinations of other NC commands may be restricted. Refer to the manual on each function.

21.4.5.2 Restriction on tool rotation type

- Unavailable commands (leading edge offset)

In the G41.3 mode, the following commands cannot be specified:

- G functions of group 01 other than G00 and G01

- Use with tool center point control

If cutter compensation for 5-axis machining is specified before tool center point control when cutter compensation for 5-axis machining and tool center point control are specified together, tool center point control must be canceled before cutter compensation for 5-axis machining is canceled (Example 1). Contrariwise, if tool center point control is specified before cutter compensation for 5-axis machining, cutter compensation for 5-axis machining must be canceled before tool center point control is canceled (Example 2).

Example 1	Example 2
G41.2 D1	G43.4 H1
:	:
G43.4 H1	G41.2 D1
:	:
G49	G40
:	:
G40	G49

If the specification sequence of G40 and G49 is reversed, alarm PS5460 is issued.

When cutter compensation for 5-axis machining is specified before tool center point control, the block for canceling tool center point control suppresses buffering. Note that, as a result, the block before the G49 block generates a compensation vector for cutter compensation for 5-axis machining, which is perpendicular to the movement.
For each of cutter compensation for 5-axis machining and tool center point control for 5-axis machining, two commands, type 1 and type 2, are provided. Be sure to specify commands of the same type. If commands of different types are specified, alarm PS5460 is issued.
When the tool is tilted by address Q in type 2 command specification, if Q is specified in both of the block for starting tool center point control for 5-axis machining and the block for starting cutter compensation for 5-axis machining, the Q command specified earlier becomes valid.

21.4.5.3 Restriction on machine configurations having table rotation axes (table rotation type and mixed-type)

- Unavailable commands

For machines having table rotation axes, the following commands cannot be specified during cutter compensation for 5-axis machining:

- G functions of group 01 other than G00 and G01

- Use with tool center point control

When the workpiece coordinate system is used as the programming coordinate system, the same restrictions as for the tool rotation type apply. See "Use with tool center point control" in the restrictions on the tool rotation type described previously.
When the table coordinate system is used as the programming coordinate system, the restrictions to apply will be explained below.

- Restrictions when the table coordinate system is used as the programming coordinate system

Use with tool center point control

When cutter compensation for 5-axis machining and tool center point control for 5-axis are specified together, tool center point control must be specified before cutter compensation. In addition, tool center point control must be canceled after cutter compensation for 5-axis machining is canceled. (Example 1)
If tool center point control is specified in the cutter compensation mode for 5-axis machining (Example 2), or if tool center point control is canceled without canceling cutter compensation for 5-axis machining (Example 3), alarm PS5460 is issued.

Example 1 (Correct specification)	Example 2 (Specification resulting in alarm)	Example 3 (Specification resulting in alarm)
G43.4 H1 : G41.2 D1 : G40 : G49	G41.2 D1 : G43.4 H1 :	G43.4 H1 : G41.2 D1 : G49

If the setting of the programming coordinate system differs between cutter compensation for 5-axis machining and tool center point control for 5-axis machining, specifying both functions together results in alarm PS5460. (See the following table:)

	TBP=0		TBP=1	
	WKP=0	WKP=1	WKP=0	WKP=1
Tool center point control for 5-axis machining	Table coordinate system	Workpiece coordinate system	Table coordinate system	Workpiece coordinate system
Cutter compensation for 5-axis machining	Workpiece coordinate system			
Both specified together	Alarm PS5460			

Programming coordinate system determined by bit 4 (TBP) of parameter No. 19746 and bit 5 (WKP) of parameter No. 19696

The table rotation axis position at the start of tool center point control for 5-axis machining and the table rotation axis position at the start of cutter compensation for 5-axis machining must always match.
Each programming coordinate system matches the workpiece coordinate system used when each function is started.
If these functions are started when their table rotation axis positions differ, the programming coordinate system for tool center point control for 5-axis machining and the programming coordinate system for cutter compensation for 5-axis machining do not match, which results in alarm PS5460.
Example: When the A-axis is the table rotation axis:

(Correct example)

```
G90 G00 A0.0
G43.4 H1
G01 Z100.0 F1000.
G41.2 D1    ← After G43.4 is specified, G41.2 is specified
            without A-axis movement.
 :
```

(Wrong example)

```
G90 G00 A0.0
G43.4 H1
G01 Z100.0 A30.0 F1000.
G41.2 D1    ← After G43.4 is specified, A-axis movement is
            made, then G41.2 is specified.
 :
```

* Furthermore, no A-axis command can be included in the block specifying G41.2.

For each of cutter compensation for 5-axis machining and tool center point control for 5-axis machining, two commands, type 1 and type 2, are provided. Be sure to specify commands of the same type. If commands of different types are specified, alarm PS5460 is issued.
When the tool is tilted by address Q in type 2 command specification, if Q is specified in both of the block for starting tool center point control for 5-axis machining and the block for starting cutter

compensation for 5-axis machining, the Q command specified earlier becomes valid.

Deceleration at a corner

Under cutter compensation for 5-axis machining, the controlled point may move along a curve even if a straight-line command is issued. Some commands may cause a corner movement.
For this reason, the tool may decelerate if small values are set as the permissible speed difference in a corner (parameter No. 1783) and permissible acceleration (parameter Nos. 1660 and 1737).

Specifiable G codes

When the table coordinate system is used as the programming coordinate system, the G codes that can be specified in the cutter compensation mode for 5-axis machining are listed below.
Specifying a G code other than these codes results in alarm PS5460:

- Positioning (G00)
- Linear interpolation (G01)
- Dwell (G04)
- Exact stop (G09)
- Programmable data input (G10)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Stored stroke check function (G22/G23)
- Cutter or tool nose radius compensation : preserve vector (G38)
- Cutter or tool nose radius compensation : corner circular interpolation (G39)
- Cutter compensation : cancel (G40)
- Tool length compensation cancel (G49)
- Scaling (G50/G51)
- Exact stop mode (G61)
- Automatic corner override mode (G62)
- Cutting mode(G64)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Absolute programming (G90)
- Incremental programming (G91)

Modal G codes that allow specification of cutter compensation for 5-axis machining

When the table coordinate system is used as the programming coordinate system, cutter compensation for 5-axis machining can be specified in the modal G code states listed below.
In a modal state other than the following modal G codes, specifying tool center point control results in alarm PS5421.

- Modal G codes included in "Specifiable G codes" described previously
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Polygon turning cancel (G50.2)
- Workpiece coordinate system 1 selection (G54 to G59)
- Canned cycle cancel (G80)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Coordinate system rotation start or 3-dimensional coordinate conversion mode on (G69)
- Feed per minute (G94)
- Polar coordinate interpolation mode cancel (G113)

T

- Mirror image for double turret off/balanced cutting mode cancel (G69)
- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69.1)
- Feed per minute (G98 (G94))

Specification of axes not relating to cutter compensation for 5-axis machining

Axes not relating to cutter compensation for 5-axis machining cannot be specified. If such an axis is specified, alarm PS5460 is issued.

21.4.6 Examples

O100 is a sample program.
This is an example in which each side of a square is cut at an angle of 30 degrees on the B-axis in a mixed-type machine.
Programs 1 to 3 all perform the same machining.

Program 1: Type 1 and the table coordinate system is selected as a programming coordinate system

```
O100（Sample Program1） ;
N10 G55 ;                              Preparations for the programming coordinate system
N20 G90 X0 Y0 Z300.0 B0 C0 ;           Movement to the initial position
N30 G01 G43.4 H01 Z40.0 F500. ;        Start of tool center point control
                                       H01 is a tool length compensation number
N40 G41.2 D01                          Start of cutter compensation
                                       D01 is a tool cutter compensation number
N50 X50.0 Y50.0 Z20.0 B30.0 C45.0 ;    The Z-axis height on the machining plane is 20.0.
N60 X-50.0 C135.0 ;
N70 X-100.0 Y-100.0 C225.0 ;
N80 X100.0 C315.0 ;
N90 X50.0 Y50.0 C405.0 ;
N100 X0 Y0 Z40.0 B0 C360.0 ;
N110 G40                               Cutter compensation cancellation
N120 G49 Z300.0 ;                      Tool center point control cancellation
                                       Movement to the initial position on the Z-axis
N130 M30;
```

Program 2: Type 1 and the workpiece coordinate system is selected as a programming coordinate system
(Note that the values specified in N50 to N90 differ from those in program 1.)

```
O100（Sample Program2）;
N10 G55 ;                              Preparations for the programming coordinate system
N20 G90 X0 Y0 Z300.0 B0 C0 ;           Movement to the initial position
N30 G01 G43.4 H01 Z40.0 F500. ;        Start of tool center point control
                                       H01 is a tool length compensation number
N40 G41.2 D01                          Start of cutter compensation
                                       D01 is a tool cutter compensation number
N50 X70.711 Y0 Z20.0 B30.0 C45.0 ;     The Z-axis height on the machining plane is 20.0.
N60 C135.0 ;
N70 X141.421 C225.0 ;
N80 C315.0 ;
N90 X70.711 C405.0 ;
N100 X0 Y0 Z40.0 B0 C360.0 ;
N110 G40                               Cutter compensation cancellation
N120 G49 Z300.0 ;                      Tool center point control cancellation
                                       Movement to the initial position on the Z-axis
N130 M30;
```

Program 3: When the type 2 is used:
(The table coordinate system is selected as a programming coordinate system)

```
O100（Sample Program3）;
N10 G55 ;                                        Preparations for the programming coordinate system
N20 G90 X0 Y0 Z300.0 B0 C0 ;                     Movement to the initial position
N30 G01 G43.5 H01 Z40.0 F500. ;                  Start of tool center point control
                                                 H01 is a tool length compensation number
N40 G41.6 D01                                    Start of cutter compensation
                                                 D01 is a tool cutter compensation number
N50 X50.0 Y50.0 Z20.0 I35.355 J35.355 K86.603 ;  The Z-axis height on the machining plane is 20.
N60 X-50.0 I-35.355 J35.355 K86.603 ;
N70 X-100.0 Y-100.0 I-35.355 J-35.355 K86.603 ;
N80 X100.0 I35.355 J-35.355 K86.603 ;
N90 X50.0 Y50.0 I35.355 J35.355 K86.603 ;
N100 X0 Y0 Z40.0 K1.0 ;
N110 G40                                         Cutter compensation cancellation
N120 G49 Z300.0 ;                                Tool center point control cancellation
                                                 Movement to the initial position on the Z-axis
N130 M30;
```

By using type 2 as in program 3, the same program can be used with machines with different configurations, regardless of whether the machine configuration is the tool rotation type, table rotation type, or mixed-type.

Composite type (tool rotation axis: B-axis - table rotation axis: C-axis - tool axis: Z direction

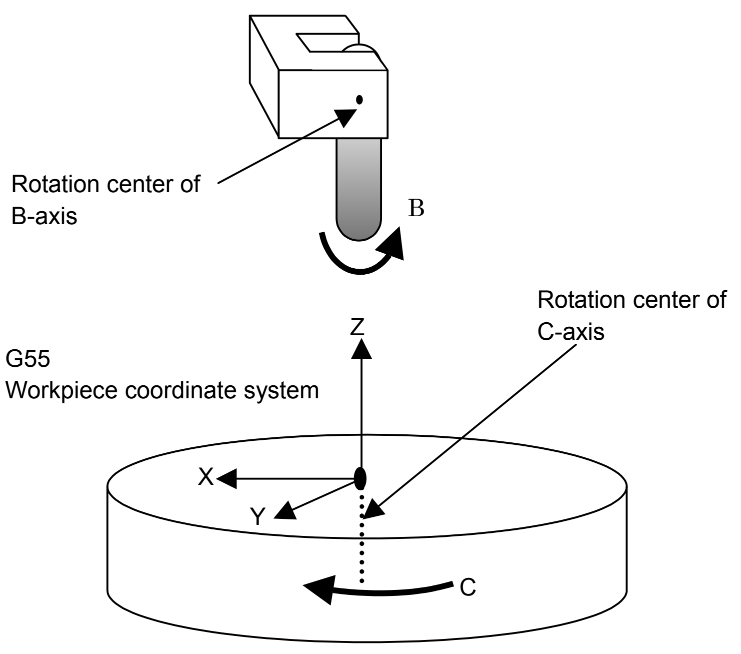


Fig. 21.4.6 (a) Machine configuration for example

Fig. 21.4.6 (b) shows the attitudes of the workpiece (object to be machined) and the tool head (relative to the workpiece (object to be machined)) as viewed in the positive Z direction of the programming coordinate system fixed to the table (table coordinate system).

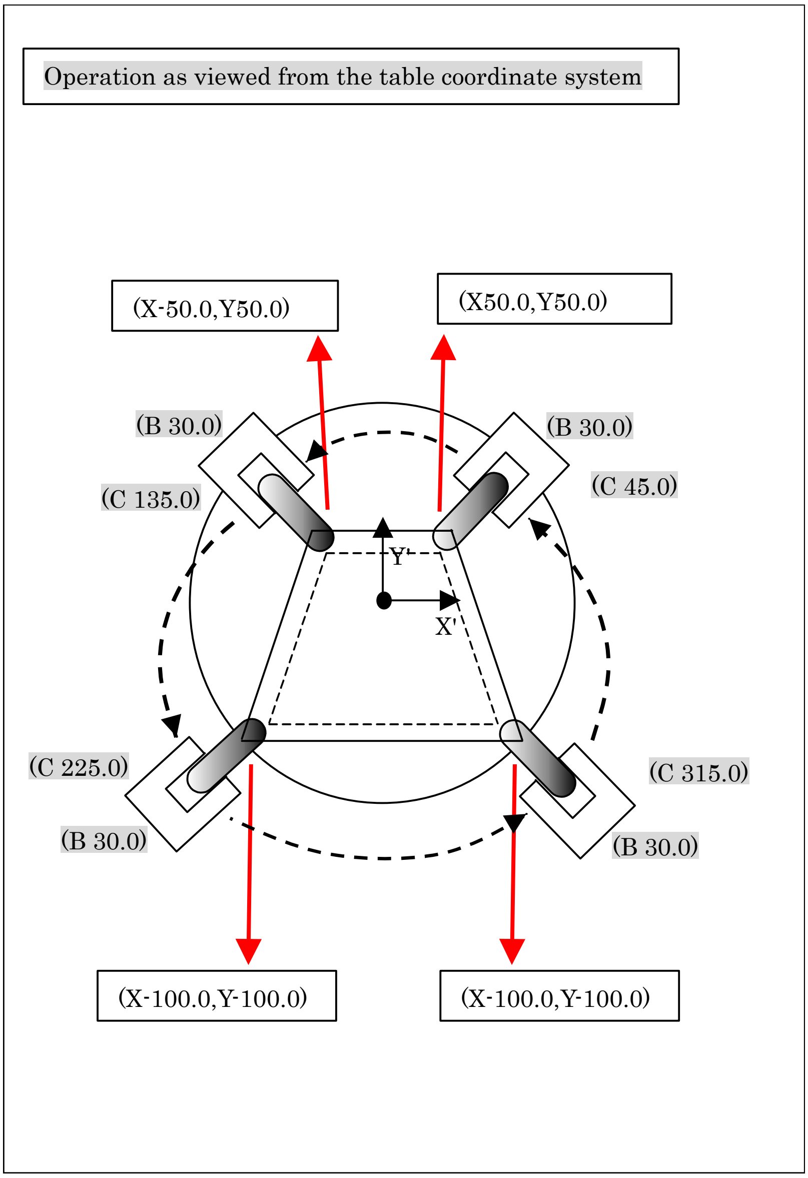


Fig. 21.4.6 (b) Illustration of example

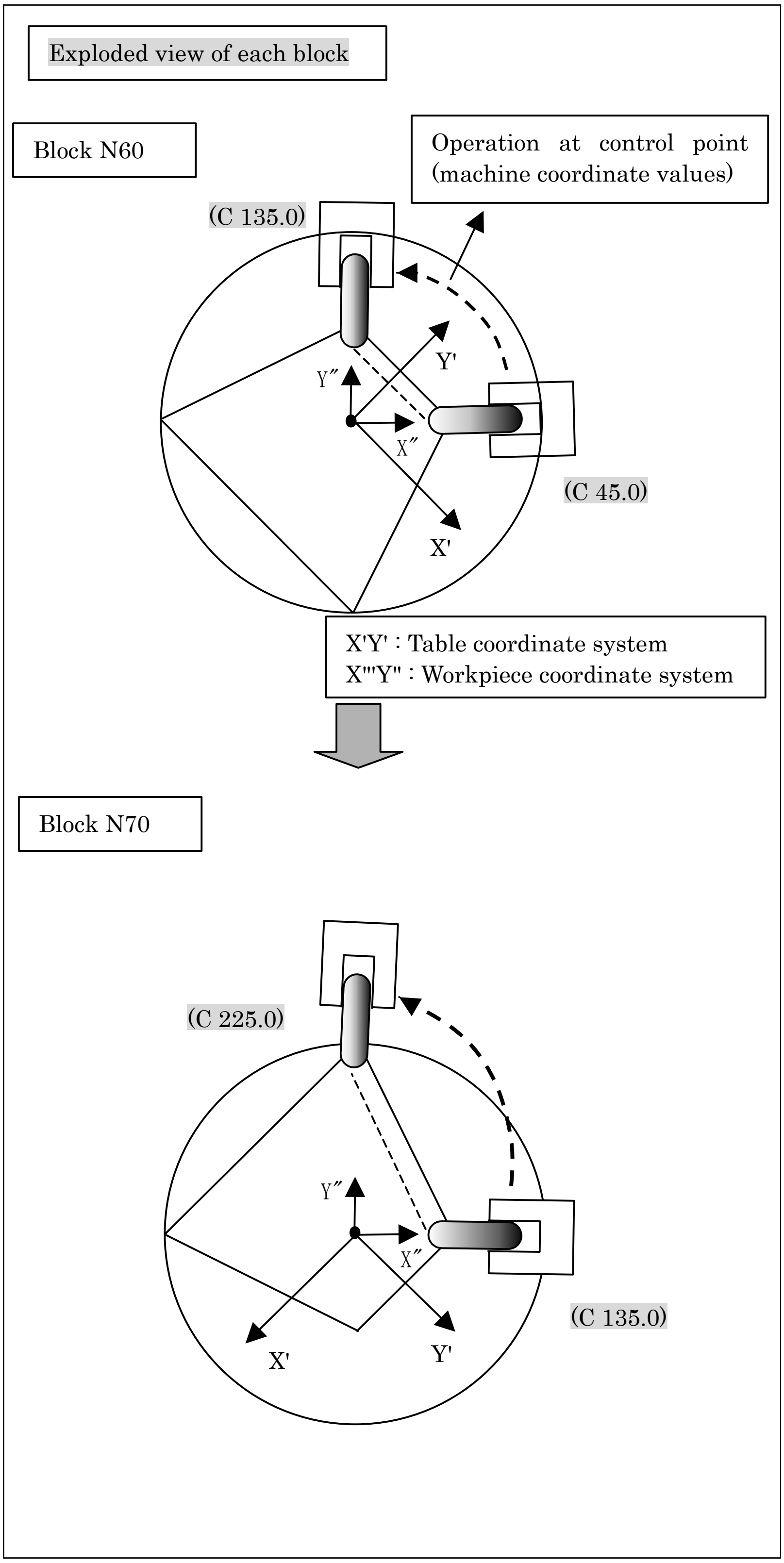


Fig. 21.4.6 (c) Exploded View of Each Block (1)

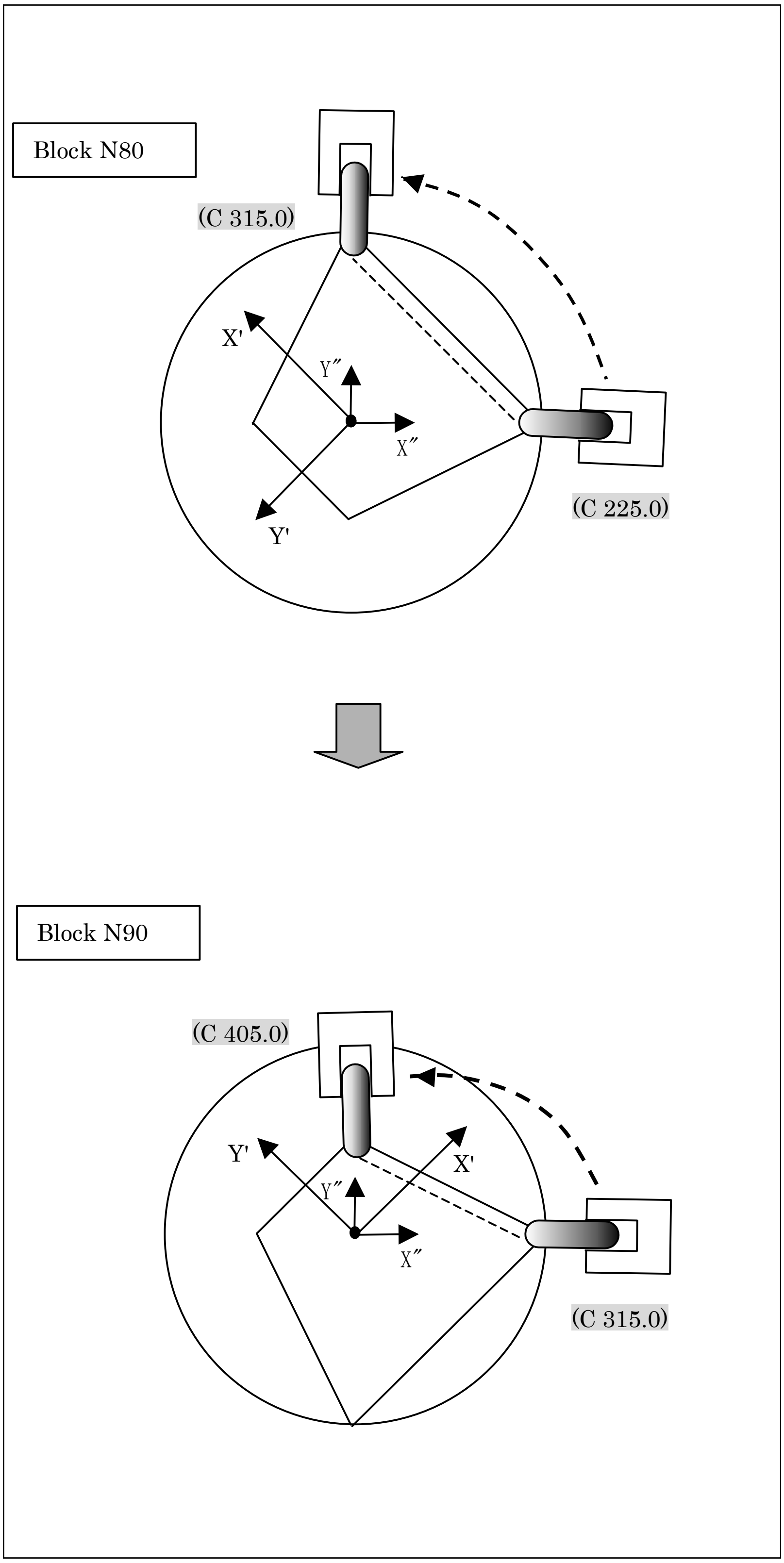


Fig. 21.4.6 (d) Exploded View of Each Block (2)

22 MUITI-PATH CONTROL FUNCTION

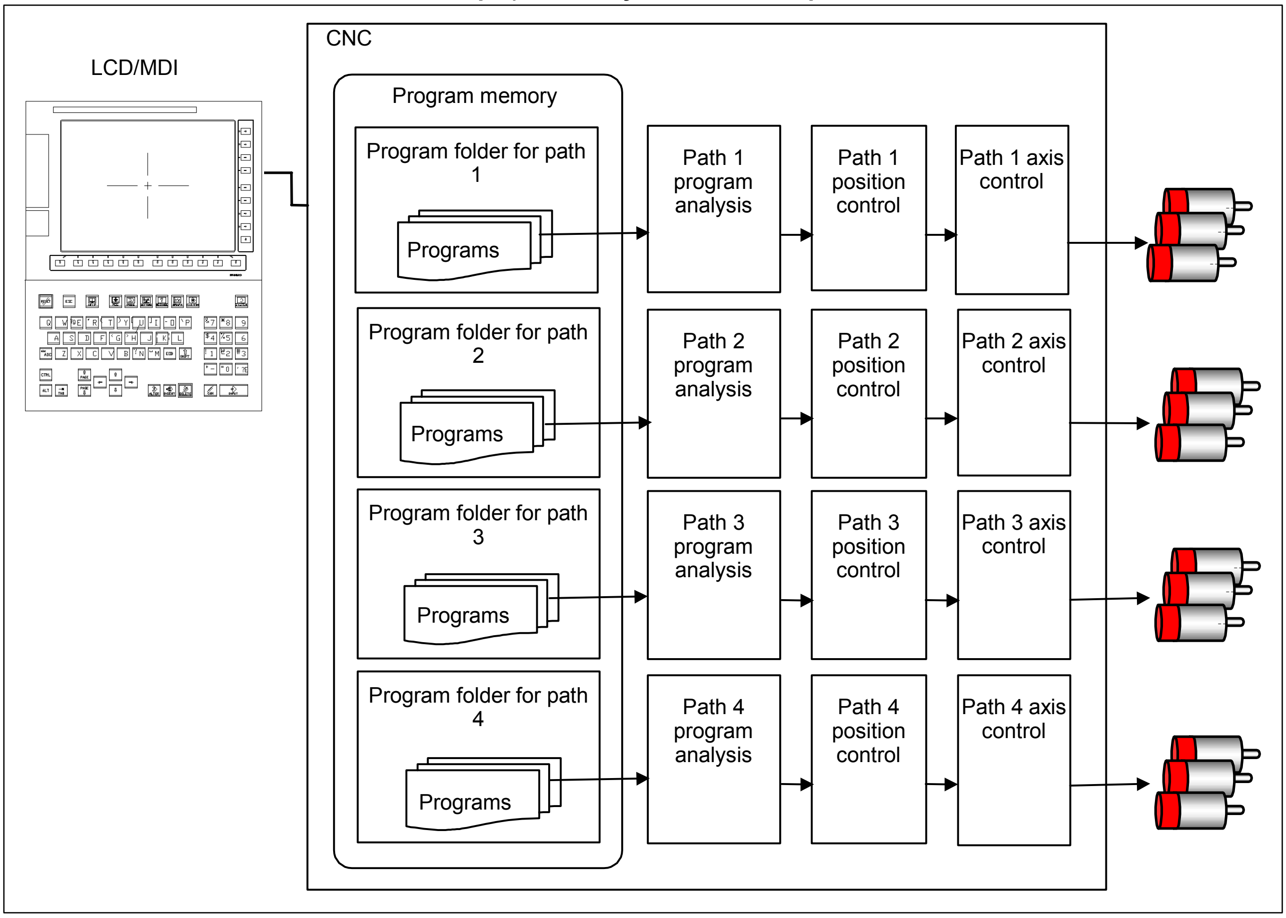
22.1 OVERVIEW

The multi-path control function is designed to enable 10 independent simultaneous machining with up to 10 paths (10-path control). This function is applicable to lathes and automatic lathes which perform cutting simultaneously with multiple tool posts, combined machine tools which perform turning and milling simultaneously with multiple paths, and machines which require additional control paths such as a loader control path.

For multi-path simultaneous machining, each machining program is stored in a folder in program memory for each path. When automatic operation is to be performed, each path is activated after selecting a program for machining with path 1 and programs for machining with paths 2 to 10 from the programs stored in the respective folders in program memory. Then the programs selected for the tool posts are executed independently at the same time. When tool post 1 and tool post 2 need to wait for each other during machining, the waiting function is available.
Other available functions specific to multi-path control include interference check for each path, balance cut, synchronous control, mixture control, spindle control between each path, and common memory between each path.

Just one LCD/MDI is provided for the all paths. Before operation and display on the LCD/MDI, the path selection signal is used to switch between the paths.

Example) For a system with four paths



22.2 WAITING FUNCTION FOR PATHS

Overview

Control based on M codes is used to cause one path to wait for the other during machining. When an M code for waiting is specified in a block for one path during automatic operation, the other path waits for the same M code to be specified before staring the execution of the next block.
A range of M codes used as M codes for waiting is to be set in the parameters (Nos. 8110 and 8111) beforehand. Waiting can be ignored using a signal.

Format

M m (Pp) ;

m: Number of an M code for waiting

p: (1) In the binary value specification mode, specify the sum of the binary values corresponding to the numbers of paths which are to wait for one another.

(2) In the path number specification mode, specify the numbers of all paths that are to wait for one another in combination.

Explanation

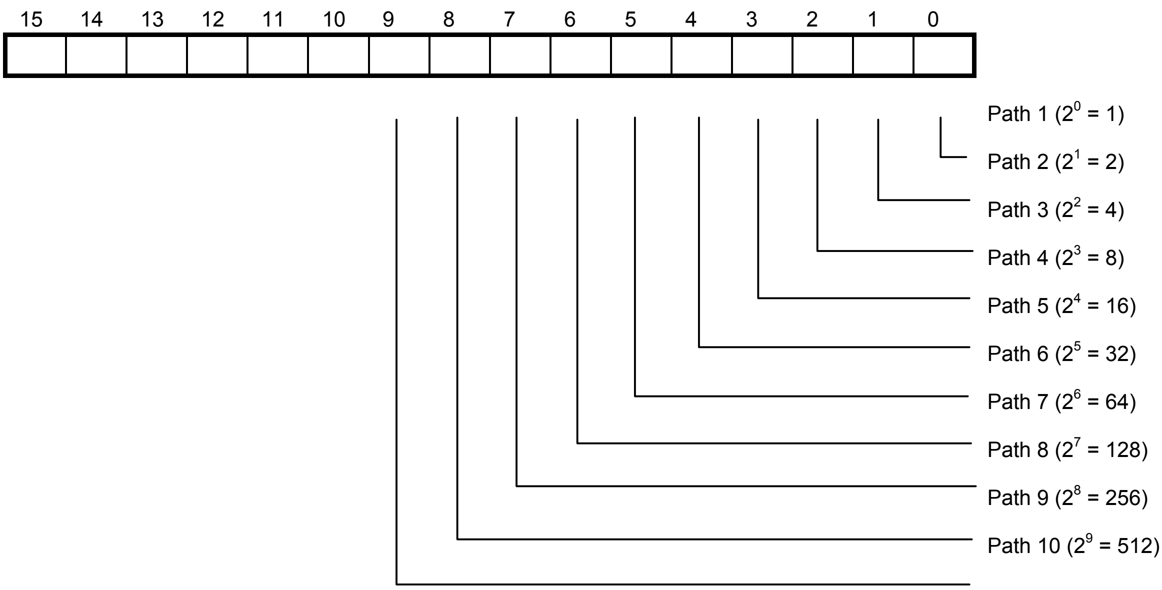
The following two methods for specifying paths which are to wait for one another at address P in the same block in which an M code for waiting is specified are available and can be selected using bit 1 (MWP) of parameter No. 8103. One method is to specify the paths with the sum of their corresponding binary values (waiting set for three or more paths specified with binary values). The other is to specify them with their path numbers in combination (waiting specified with path numbers).
When address P is not specified, paths 1 and 2 wait for each other (waiting set for two paths). Always specify address P in a single block.

- Waiting specified with binary values

When bit 1 (MWP) of parameter No. 8103 is set to 0, the value specified at address P is assumed to be obtained using binary values. The following table lists the path numbers and corresponding binary values.

Path number	Binary value (decimal number)
1	1
2	2
3	4
4	8
5	16
6	32
7	64
8	128
9	256
10	512

The bit position of each path in binary representation is shown below.



To make all of paths 1, 2, and 3 wait for one another, the P value is obtained as follows:

Binary value of path 1	1 (0000 0000 0000 0001)
Binary value of path 2	2 (0000 0000 0000 0010)
Binary value of path 3	4 (0000 0000 0000 0100)
Sum	7 (0000 0000 0000 0111)

All of the three paths can be made to wait for one another by specifying P7 together with an M code for waiting.
To make all of paths 1, 3, 5, 7, and 9 wait for one another, the P value is obtained as follows:

Binary value of path 1 1 (0000 0000 0000 0001)

Binary value of path 3	4 (0000 0000 0000 0100)
Binary value of path 5	16 (0000 0000 0001 0000)
Binary value of path 7	64 (0000 0000 0100 0000)
Binary value of path 9	256 (0000 0001 0000 0000)
Sum	341 (0000 0001 0101 0101)

All of the five paths can be made to wait for one another by specifying P341 together with an M code for waiting.

- Waiting specified with a combination of path numbers

When bit 1 (MWP) of parameter No. 8103 is set to 1, the value specified at address P is assumed to be a combination of path numbers. The following table lists the path numbers and corresponding values.

Path number	Value (decimal number)
1	1
2	2
3	3
4	4
5	5
6	6
7	7
8	8
9	9
10	0

To make all of paths 1, 2, and 3 wait for one another, the P value is a number consisting of 1, 2, and 3.

Example) P123

There are no restrictions on the order in which the numeric characters are specified, the following six possible values can be specified:

P123, P132, P213, P231, P312, P321

Path numbers specified in combination in different orders for different paths are effective as long as the numbers of the relevant paths are specified.

Example) The following are treated as the same P value and these paths can be made to wait for one another:
M200P123 for path 1, M200P231 for path 2, and M200P321 for path 3

To make all of paths 1, 3, 5, 7, and 9 wait for one another, the P value is a number consisting of 1, 3, 5, 7, and 9.

Example) P13579

- Waiting for path 10

To make path 10 and another path wait for each other, specify a value of 0 for the combination.
If a number begins with 0, 0 cannot be recognized. Specify 0 in the second or subsequent digit from the left.

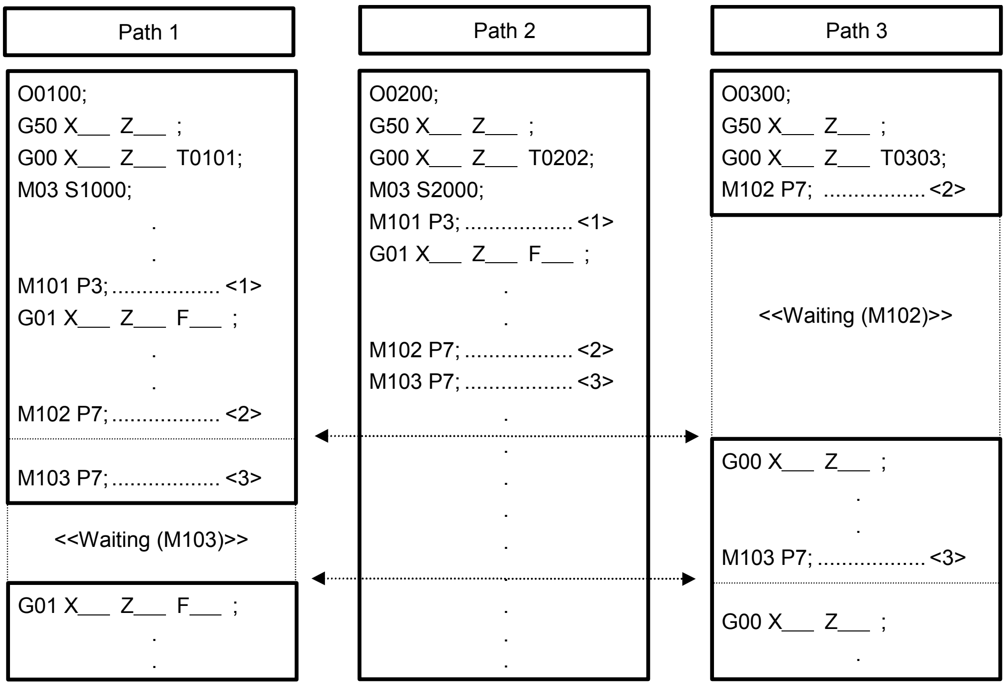
Incorrect example) P013579
Correct example) P103579

In the incorrect example, the P value is assumed to be the same as P13579. Path 10 cannot be made to wait for other paths and other paths cannot be made to wait for path 10.

Example

- When the value specified at P is obtained using binary values

Assume that the waiting ignore signal for path 2 (bit 7 of G1063 for a system with three or more paths) is set to 1 and M101 to M103 (parameter No. 8110 = 101 and parameter No. 8111 = 103) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:



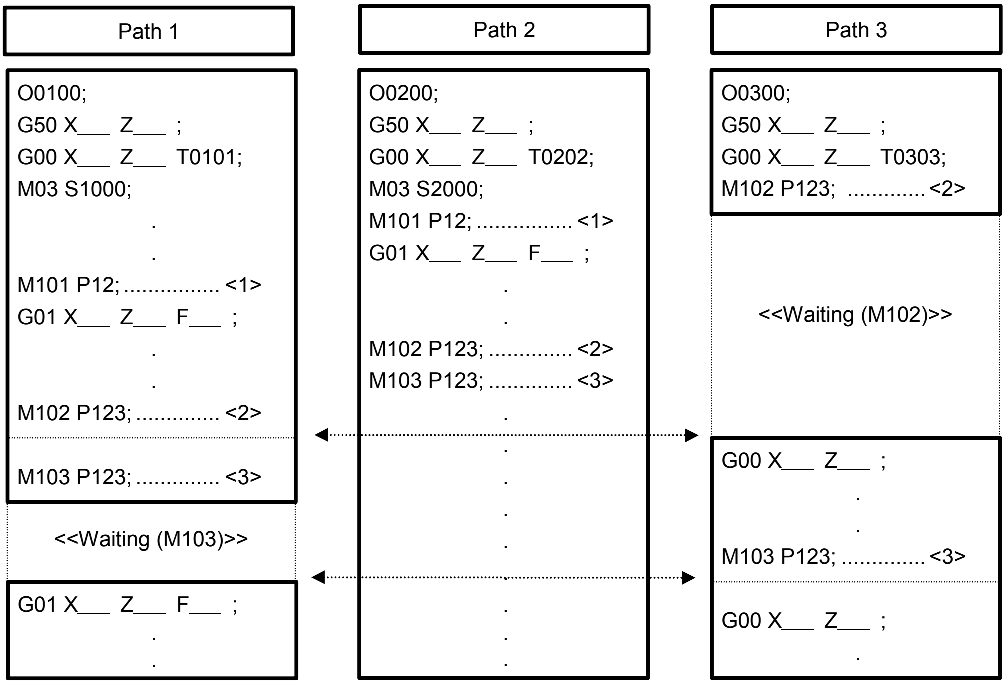
<1> M101 P3; (making paths 1 and 2 wait for each other)
When the waiting ignore signal for path 2 is set to 0, paths 1 and 2 wait for each other. Because the signal is set to 1, however, paths 1 and 2 ignore the M code for waiting and immediately execute the next block.

<2> M102 P7; (making paths 1, 2, and 3 wait for one another)
In this example, path 3 waits for processing on paths 1 and 2 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 3 assumes that waiting is completed when processing on path 1 terminates, and executes the next block.

<3> M103 P7; (making paths 1, 2, and 3 wait for one another)
In this example, paths 1 and 2 wait for processing on path 3 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 2 does not wait for processing on path 3 to terminate and executes the next block, but path 1 waits for path 3.

- When the value specified at P is obtained using path numbers in combination

Assume that the waiting ignore signal for path 2 (bit 7 of G1063 for a system with three or more paths) is set to 1 and M101 to M103 (parameter No. 8110 = 101 and parameter No. 8111 = 103) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:



<1> M101 P12; (making paths 1 and 2 wait for each other)
When the waiting ignore signal for path 2 is set to 0, paths 1 and 2 wait for each other. Because the signal is set to 1, however, paths 1 and 2 ignore the M code for waiting and immediately execute the next block.

<2> M102 P123; (making paths 1, 2, and 3 wait for one another)
In this example, path 3 waits for processing on paths 1 and 2 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 3 assumes that waiting is completed when processing on path 1 terminates, and executes the next block.

<3> M103 P123; (making paths 1, 2, and 3 wait for one another)
In this example, paths 1 and 2 wait for processing on path 3 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 2 does not wait for processing on path 3 to terminate and executes the next block, but path 1 waits for path 3.

⚠ **CAUTION**

1 An M code for waiting must always be specified in a single block.
2 Unlike other M codes, the M code for waiting is not output to the PMC.
3 If the operation of a single path is required, the M code for waiting need not be deleted. By using the signal to specify that waiting be ignored (NOWT for the system with two paths, NMWT for the system with three or more paths), the M code for waiting in a machining program can be ignored. For details, refer to the manual supplied by the machine tool builder.
4 If using a waiting M code in block multiple-M code command mode, be sure to specify it as the first M code.

22.3 COMMON MEMORY BETWEEN EACH PATH

Overview

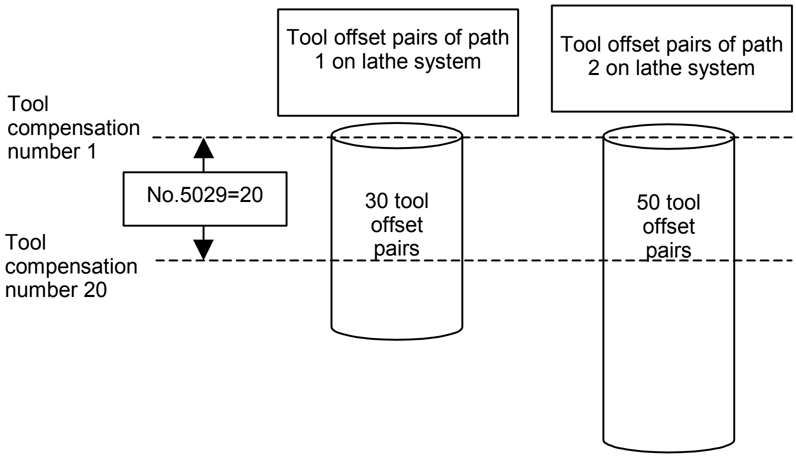
In a multi-path system, this function enables data within the specified range to be accessed as data common to all paths. The data includes tool compensation memory and custom macro common variables.

Explanation

The path common memory function enables the following operations.

- Tool compensation memory

Part or all of tool compensation memory for individual paths can be used as common data by setting parameter No. 5029.

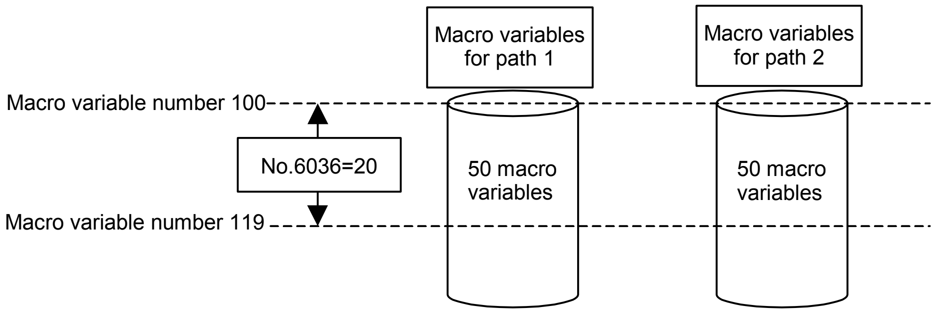


NOTE

1 When a combined system including a machining center system and lathe system is used, data is made common according to the same path control type.
2 The same unit for tool compensation (bits 0 to 3 of parameter No. 5042) must be set for each of the machining center and lathe systems.
3 Set a value less than the number of tool compensation values for each path for parameter No. 5029.
4 If the value set for parameter No. 5029 is greater than the number of tool compensation values for each path, the minimum number of tool compensation values for individual paths is assumed.
5 For details, refer to the relevant manual of the machine tool builder.

- Custom macro common variables

All or part of custom macro common variables #100 to #149 (, #199, or #499) and #500 to #599 (or #999) can be used as common data by setting parameters Nos. 6036 (#100 to #149 (, #199, or #499)) and 6037 (#500 to #599 (or #999)).



> **NOTE**
> 1 If the value of parameter No. 6036 or 6037 exceeds the maximum number of macro common variables, the maximum number of macro common variables is assumed.
> 2 Common variables #150 to #199, #150 to #499, and #600 to #999 are optional functions.

22.4 SPINDLE CONTROL BETWEEN EACH PATH

Overview

This function allows a workpiece attached to one spindle to be machined simultaneously with two tool posts and each of two workpieces attached to each of two spindles to be machined simultaneously with each of two tool posts.

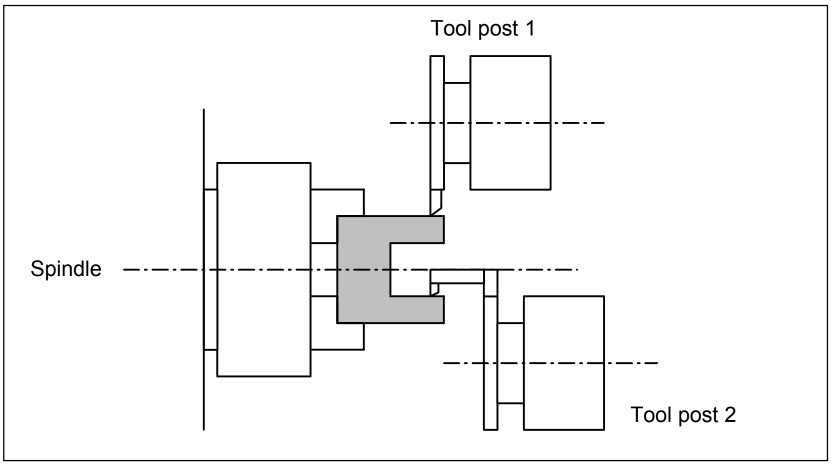


Fig. 22.4 (a) Application to a lathe with one spindle and two tool posts

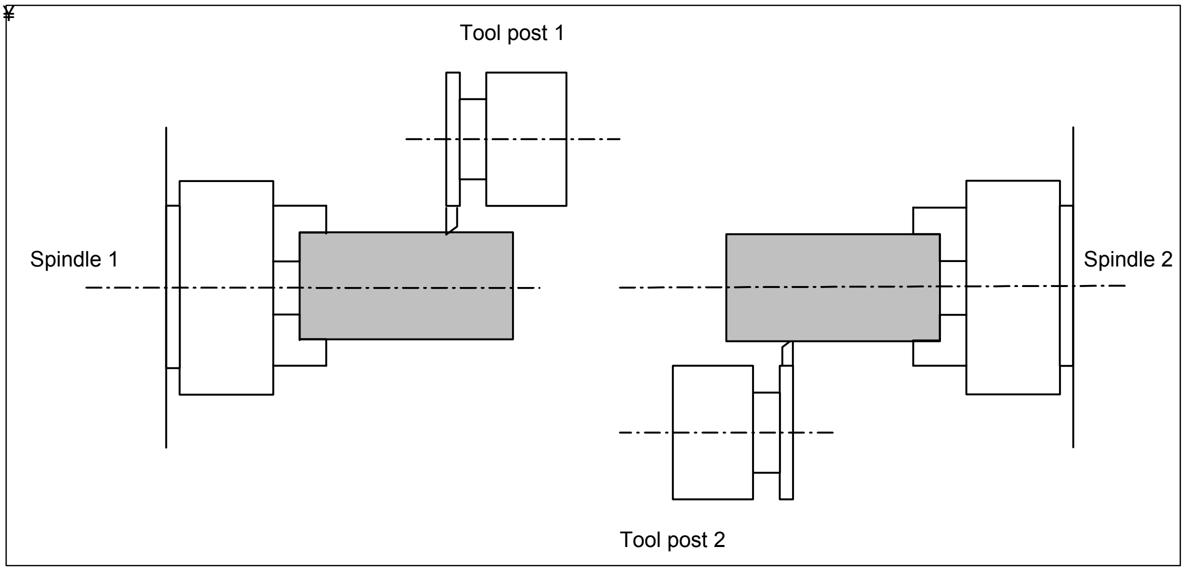


Fig. 1.1 (b) Application to a lathe with two spindles and two tool posts

The spindle belonging to each path can generally be controlled by programmed commands for the path. With path spindle command selection signals, programmed commands for any path can control the spindle belonging to any path.

> **NOTE**
> For the method of spindle command selection, refer to the relevant manual of the machine tool builder.

22.5 SYNCHRONOUS CONTROL, MIXTURE CONTROL, AND SUPERPOSITION CONTROL

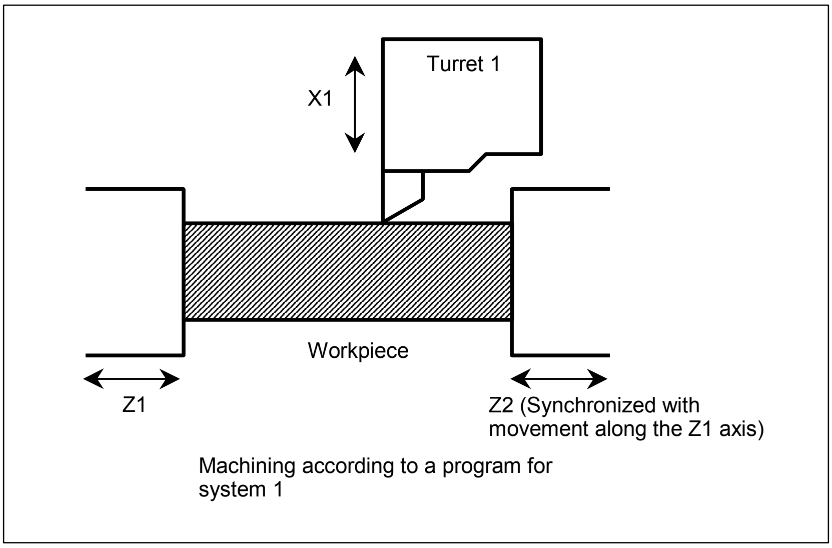
Overview

In multi-path control, the synchronous control function, mixture control function, and superimposed control function enable synchronous control, mixture control, and superimposed control in a single system or between multiple systems.

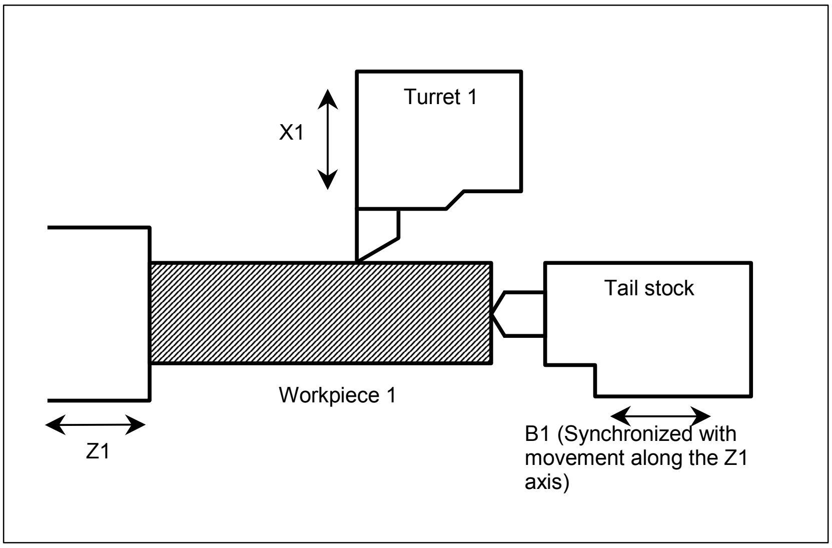
Explanation

- Synchronous control

- Synchronizes movement along an axis of one system with that along an axis of the other system.
  Example)
  Synchronizing movement along the Z1 (master) and Z2 (slave) axes (in the case of turning)



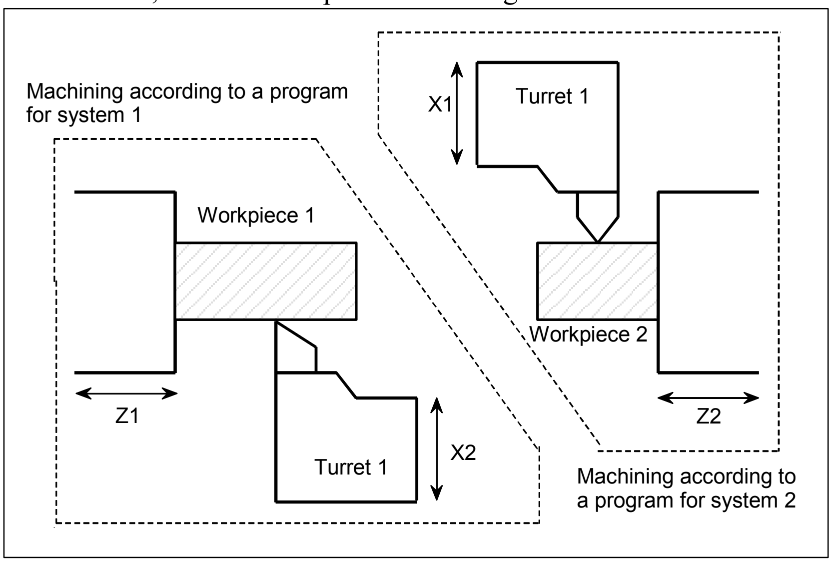
- Synchronizes movement along an axis of one system with that along another axis of the same system.
  Example)
  Synchronizing movement along the Z1 (master) and B1 (slave) axes (in the case of turning)



- Mixture control

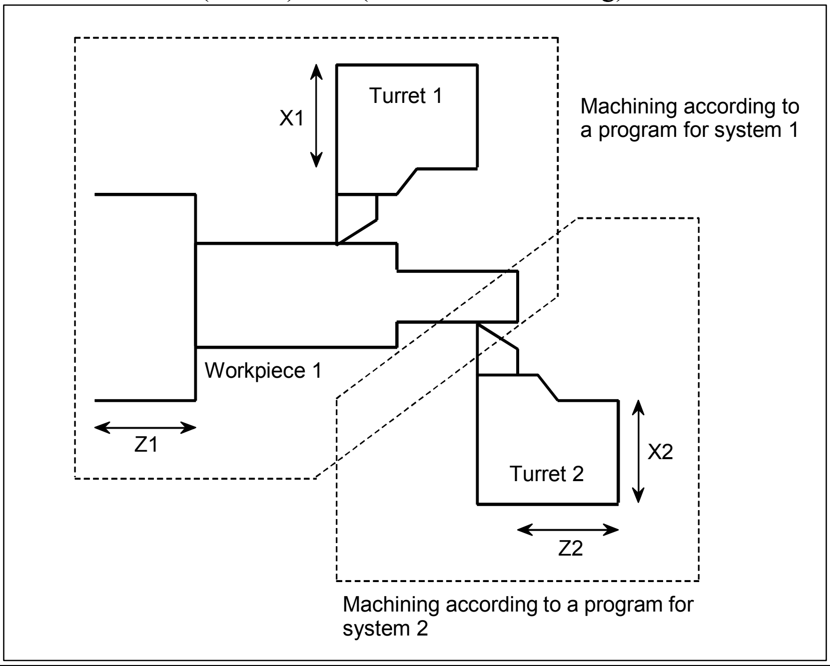
- Exchanges the move commands for different axes of different system.

  Example)
  Exchanging the commands for the X1 and X2 axes (in the case of turning)
  → Upon the execution of a command programmed for system 1, movement is performed along the X2 and Z1 axes.
  Upon the execution of a command programmed for system 2, movement is performed along the X1 and Z2 axes.



- Superimposed control

- Provides a move command of an axis for a different axis in another system.
  Example)
  Providing the Z2 (slave) axis with a move command specified for the Z1 (master) axis (in the case of turning)



> **NOTE**
> The method used to specify synchronous, mixture, or superimposed control varies with the machine tool builder. For details, refer to the manual supplied by the machine tool builder.

III. OPERATION

1 GENERAL

1.1 MANUAL OPERATION

Explanation

- Manual reference position return

The CNC machine tool has a position used to determine the machine position.
This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on, the tool is moved to the reference position.
Manual reference position return is to move the tool to the reference position using switches and pushbuttons located on the operator's panel. (See Section III-3.1)

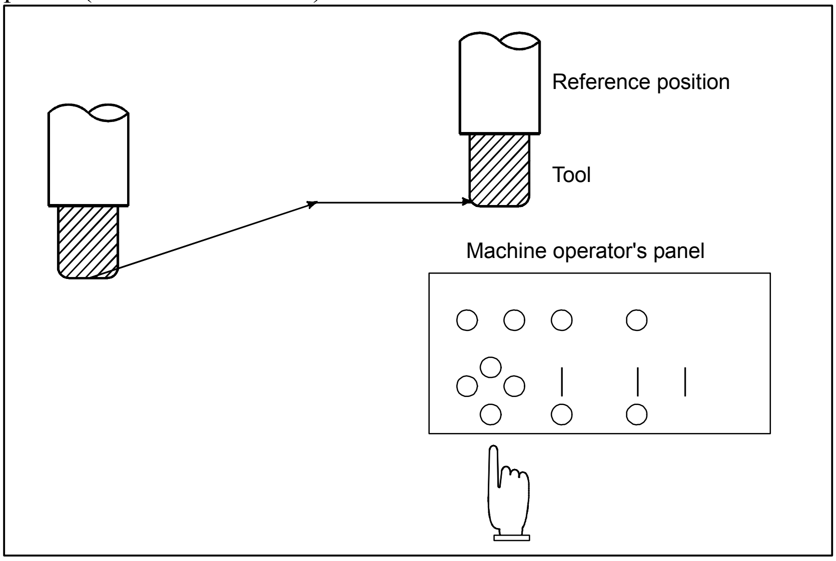


Fig. 1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.
This operation is called automatic reference position return (See Section II-6).

- The tool movement by manual operation

Using machine operator's panel switches, pushbuttons, or the manual handle, the tool can be moved along each axis.

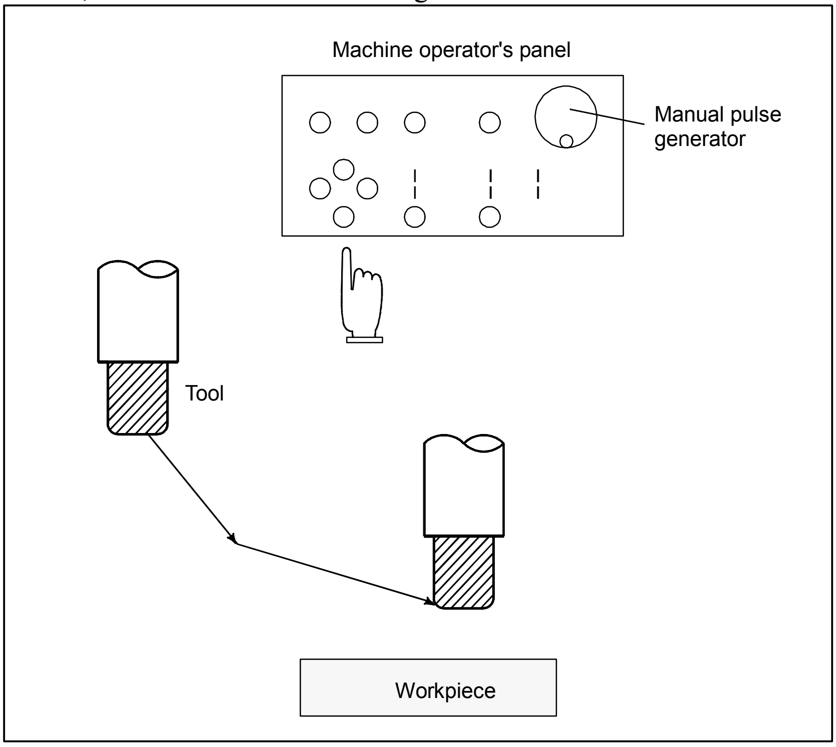


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

(i) Jog feed (See Section III-3.2)
The tool moves continuously while a pushbutton remains pressed.

(ii) Incremental feed (See Section III-3.3)
The tool moves by the predetermined distance each time a button is pressed.

(iii) Manual handle feed (See Section III-3.4)
By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

1.2 TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory, MDI and DNC operations. (See Section III-4).

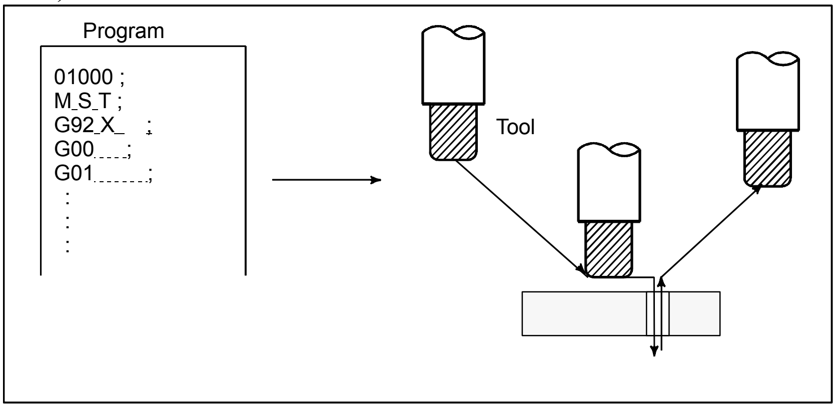


Fig. 1.2 (a) Tool Movement by programming

Explanation

- Memory operation

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation.

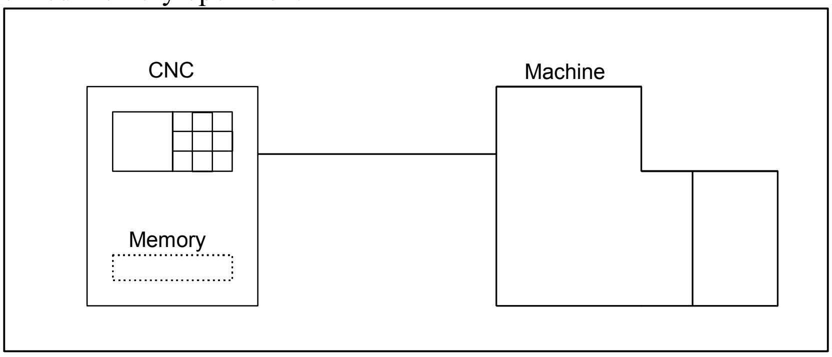


Fig. 1.2 (b) Memory operation

- MDI operation

After the program is entered, as an command group, from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation.

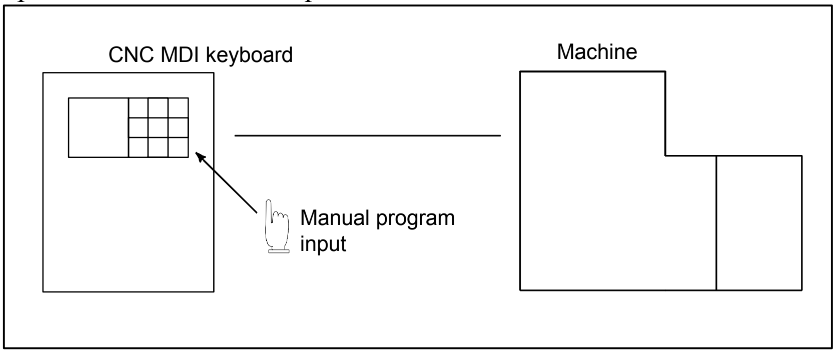


Fig. 1.2 (c) MDI operation

- DNC operation

In this mode of operation, the program is not registered in the CNC memory. It is read from the external input/output devices instead. This is called DNC operation. This mode is useful when the program is too large to fit the CNC memory.

1.3 AUTOMATIC OPERATION

Explanation

- Program selection

Select the program used for the workpiece. Ordinarily, one program is prepared for one workpiece. If two or more programs are in memory, select the program to be used, by searching the program number (Section III-9.3).

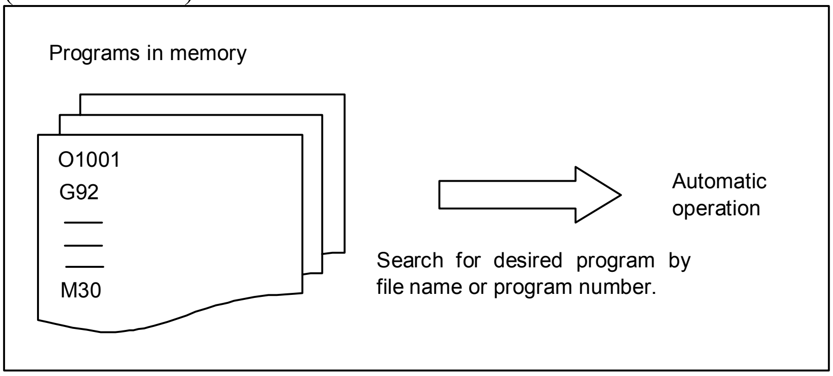


Fig. 1.3 (a) Program selection for automatic operation

- Start and stop

Pressing the cycle start pushbutton causes automatic operation to start. By pressing the feed hold or reset pushbutton, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops. (See Section III-4)

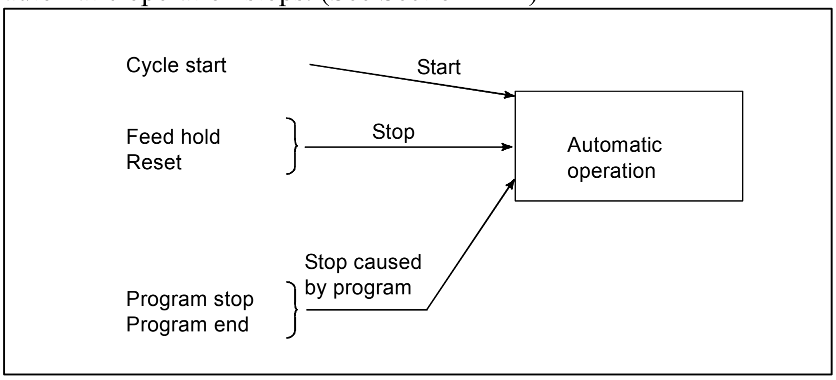


Fig. 1.3 (b) Start and stop for automatic operation

- Handle interruption

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle. (See Section III-4.4)

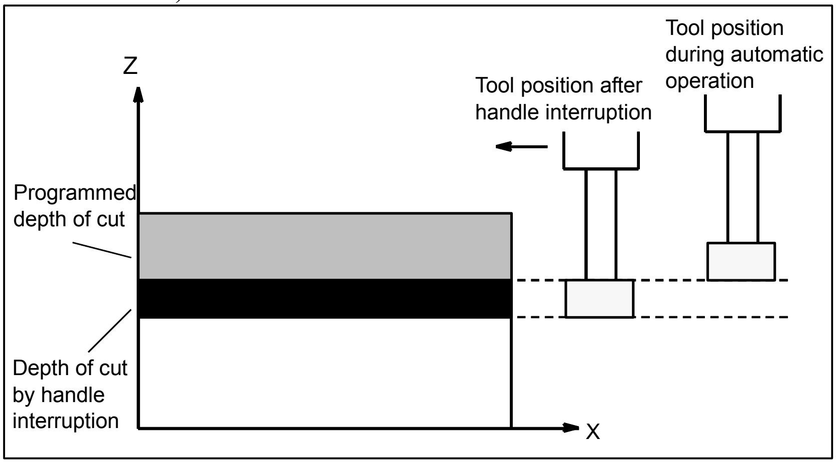


Fig. 1.3 (c) Handle interruption for automatic operation

1.4 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed.
It checks whether the created program can operate the machine as desired.
This check can be accomplished by running the machine actually or viewing the position display change (without running the machine) (See Section III-5).

1.4.1 Check by Running the Machine

Explanation

- Dry run

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel. (See Section III-5.4)

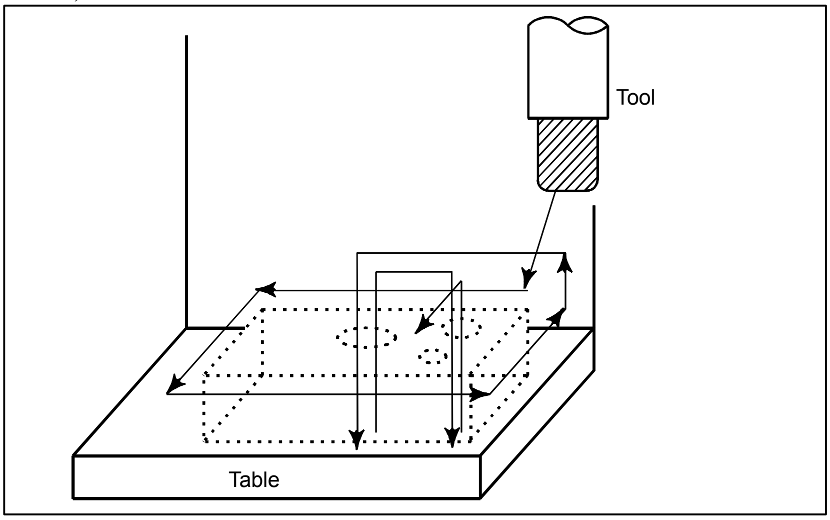


Fig. 1.4.1 (a) Dry run

- Feedrate override

Check the program by changing the feedrate specified in the program. (See Section III-5.2)

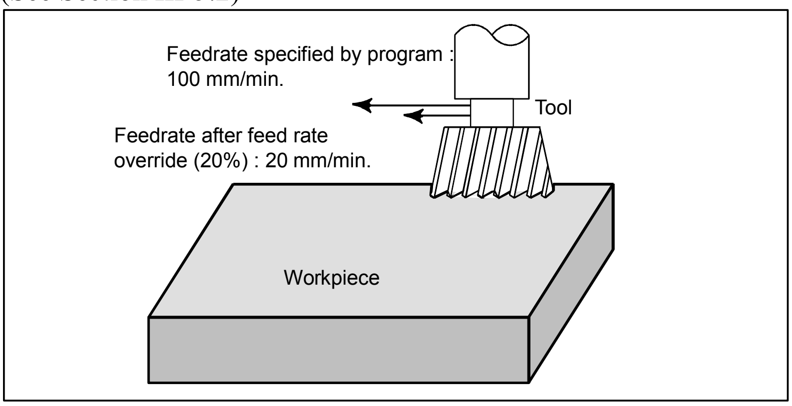


Fig. 1.4.1 (b) Feedrate override

- Single block

When the cycle start pushbutton is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner. (See Section III-5.5)

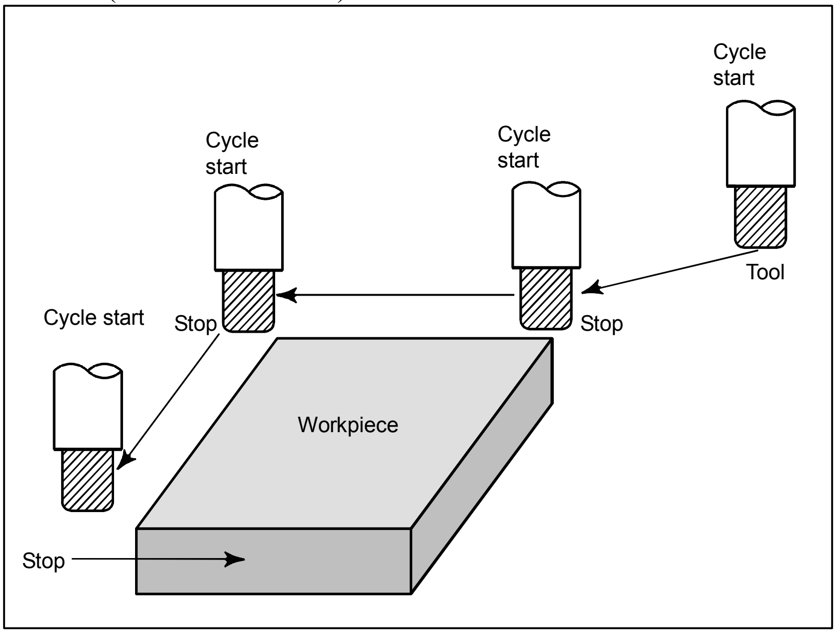


Fig. 1.4.1 (c) Single Block

1.4.2 How to View the Position Display Change without Running the Machine

Explanation

- Machine Lock

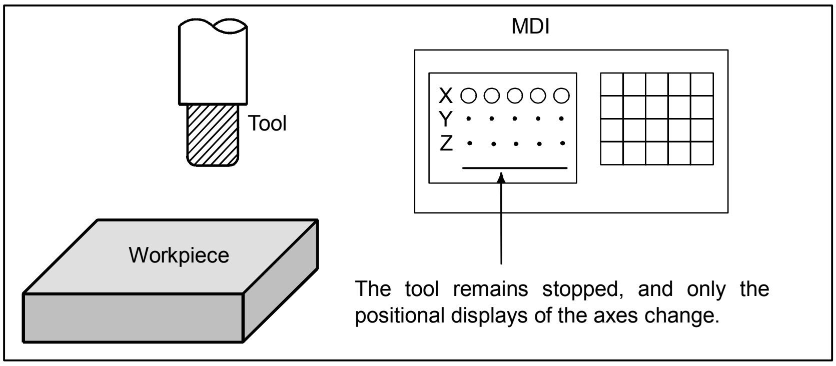


Fig. 1.4.2 (a) Machine Lock

- Auxiliary function lock

When automatic running is placed into the auxiliary function lock mode during the machine lock mode (See Sections III-5.1), all auxiliary functions (spindle rotation, tool replacement, coolant on/off, etc.) (See Section III-5.1) are disabled.

1.5 EDITING A PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the MDI panel (See Section III-10).
This operation can be executed using the program edit function.

1.6 DISPLAYING AND SETTING DATA

The operator can display or change a value stored in CNC internal memory by key operation on the MDI screen (See III-12).

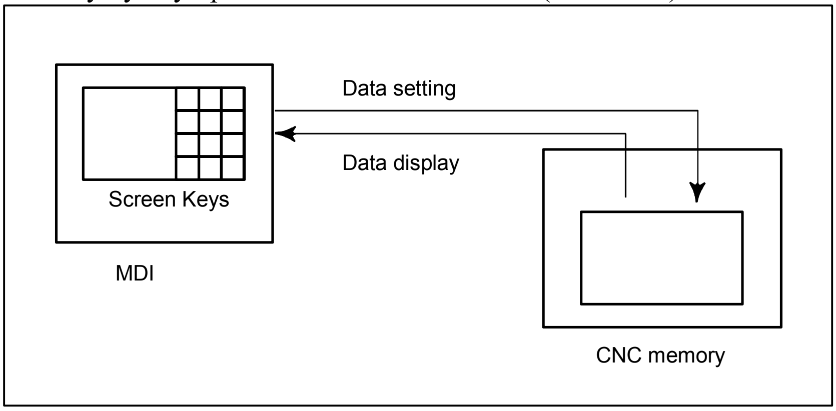


Fig. 1.6 (a) Displaying and setting data

Explanation

- Offset value

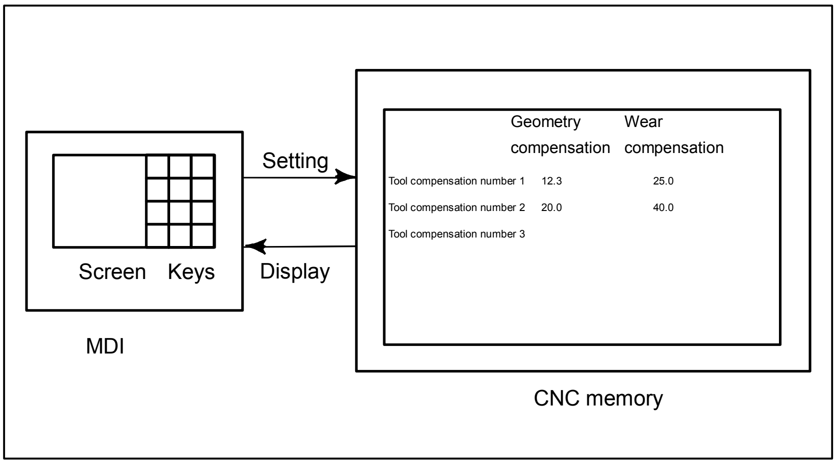


Fig. 1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement value depends on the tool dimensions. By setting tool dimension data in CNC memory beforehand, automatically generates tool routes that permit any tool to cut the workpiece specified by the program. Tool dimension data is called the offset value.

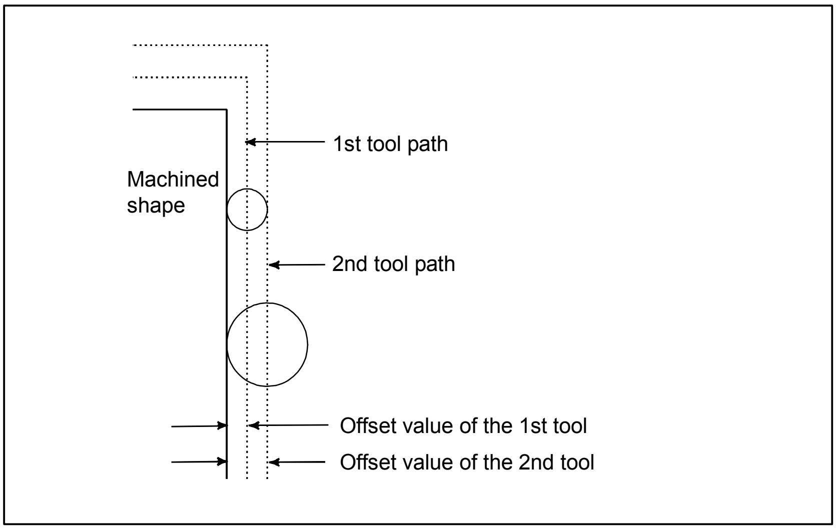


Fig. 1.6 (c) Offset value

- Displaying and setting operator's setting data

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.
For example, the following data can be set:

- Inch/Metric switching
- Selection of I/O devices
- Mirror image cutting on/off

The above data is called setting data (See Section III-12.3.1).

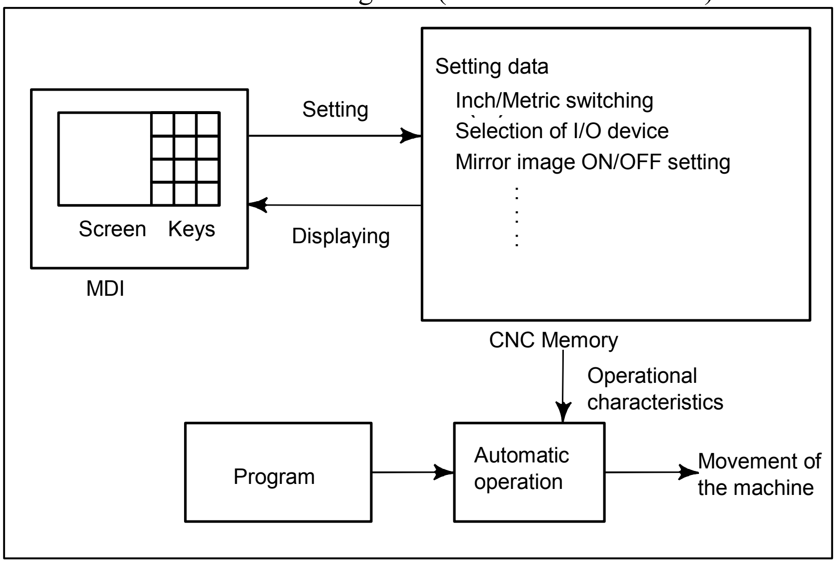


Fig. 1.6 (d) Displaying and setting operator's setting data

- Displaying and setting parameters

The CNC functions have versatility in order to take action in characteristics of various machines.
For example, CNC can specify the following:

- Rapid traverse rate of each axis
- Whether increment system is based on metric system or inch system.
- How to set command multiply/detect multiply (CMR/DMR)

Data to make the above specification is called parameters (See Section III-12.4.1).
Parameters differ depending on machine tool.

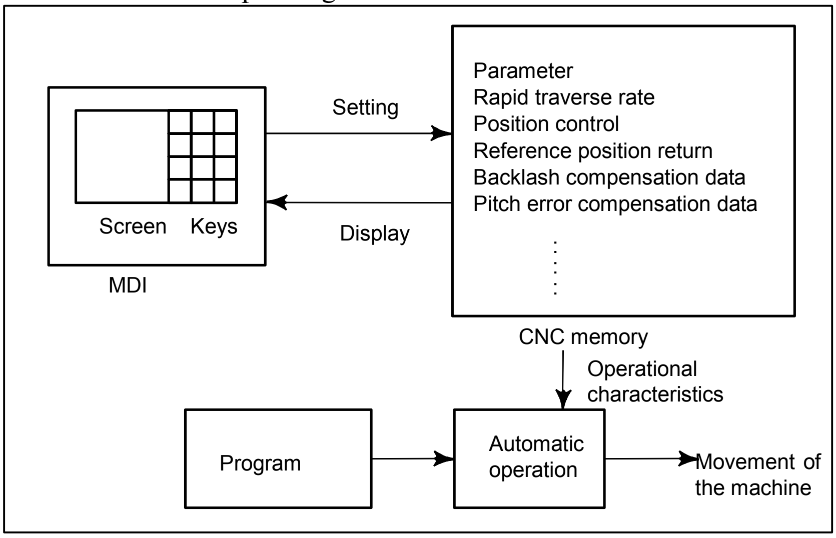


Fig. 1.6 (e) Displaying and setting parameters

- Data protection key

A key called the data protection key can be defined. It is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously (See Section III-12).

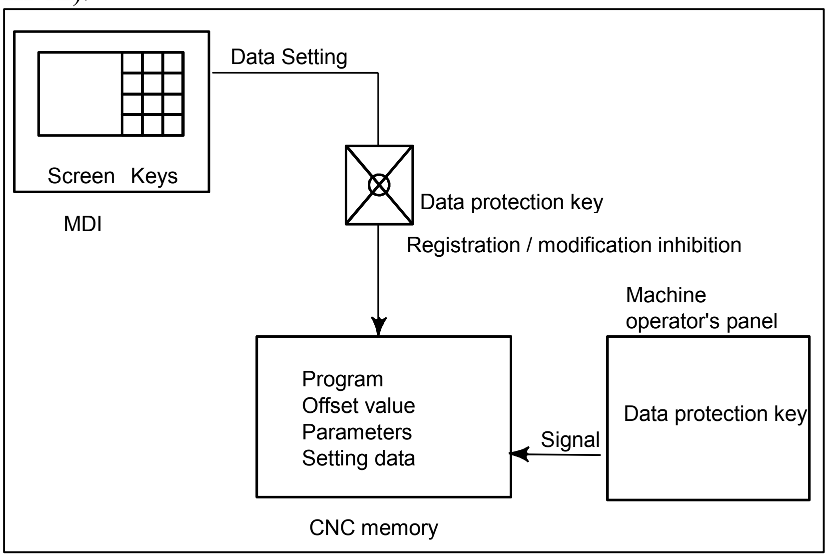


Fig. 1.6 (f) Data protection key

1.7 DISPLAY

1.7.1 Program Display

The contents of the currently active program are displayed. In addition, the
(See Section III-12.2.1)

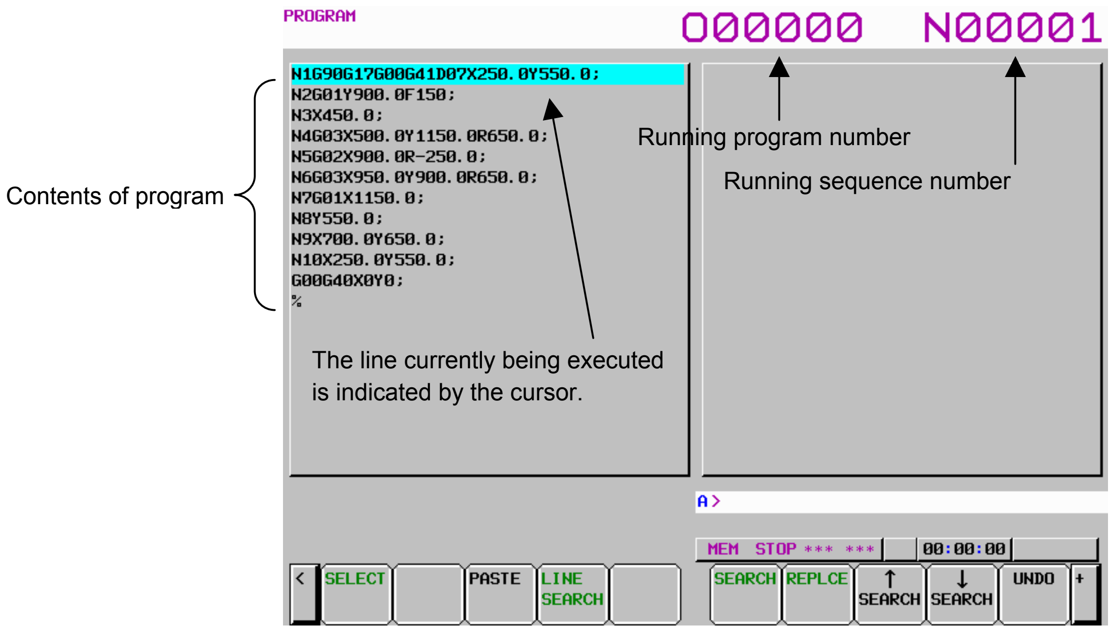


Fig. 1.7.1 (a)

A list of the programs held in the currently selected folder is displayed.

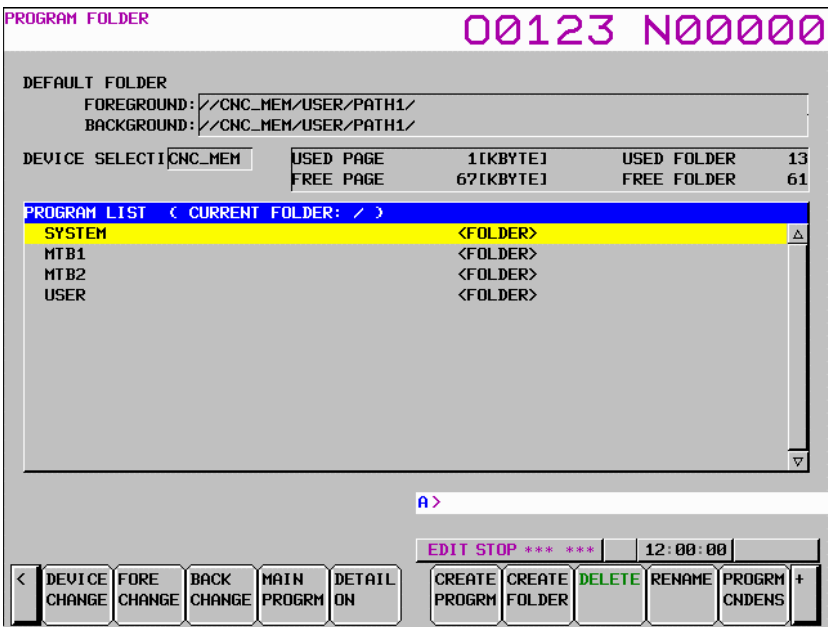


Fig. 1.7.1 (b)

1.7.2 Current Position Display

The current position of the tool is displayed with the coordinate values.
Moreover, the distance from the current position to a target point can be displayed as a remaining travel distance.
(See Section III-12.1.1, 12.1.2, 12.1.3.)

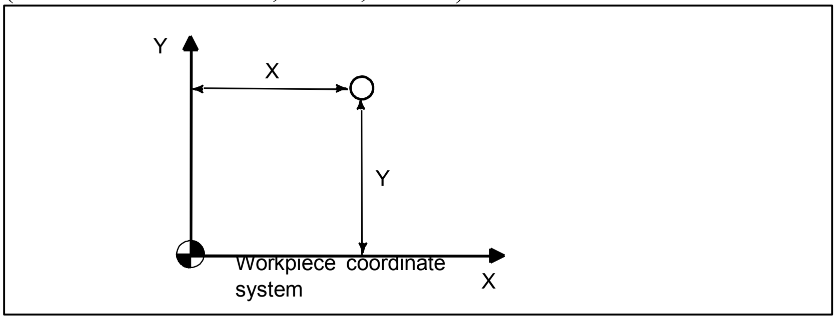


Fig. 1.7.2 (a)

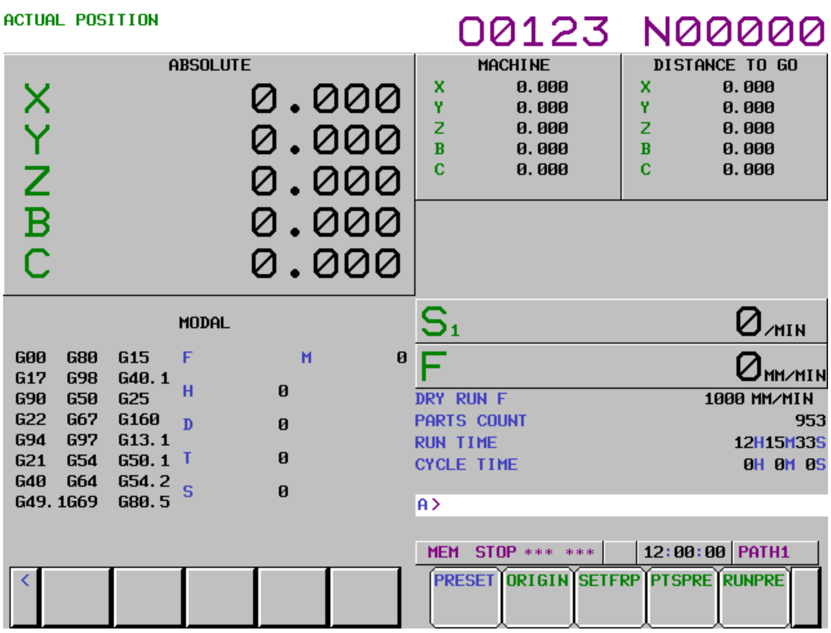


Fig. 1.7.2 (b)

1.7.3 Alarm Display

When a trouble occurs during operation, error code and alarm message are displayed on the screen. (See Section III-7.1.)
See APPENDIX G for the list of error codes and their meanings.

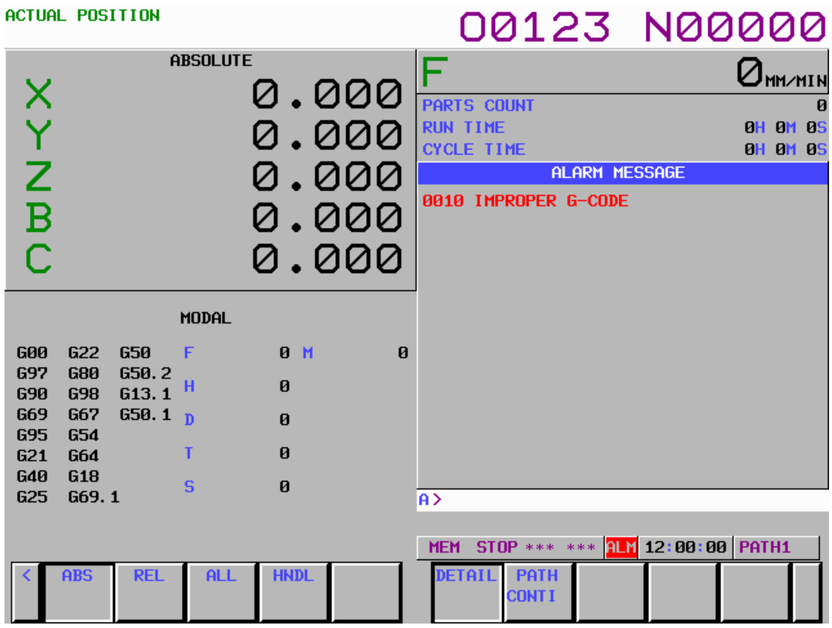


Fig. 1.7.3 (a)

1.7.4 Parts Count Display, Run Time Display

The position display screen displays a run time, cycle time, and parts count. (See Section lll-12.3.3.)

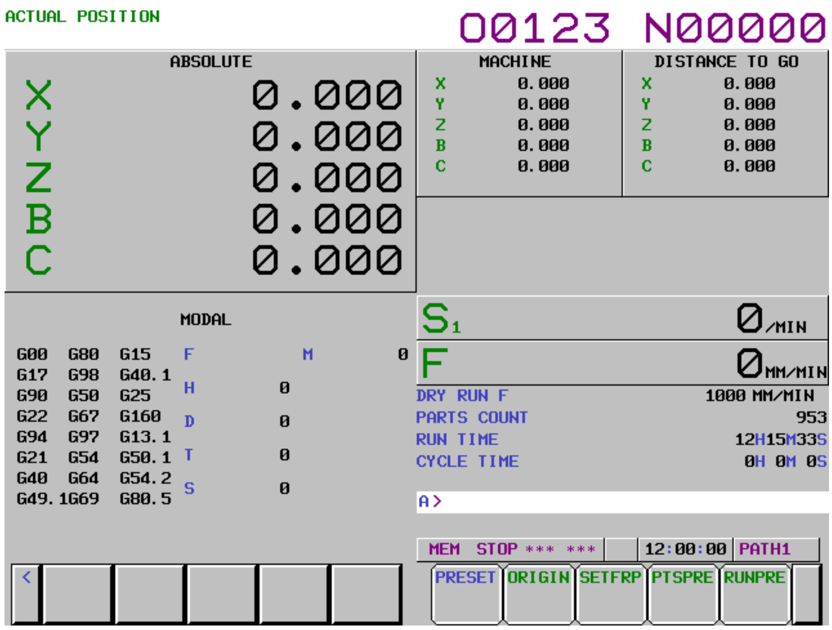


Fig. 1.7.4 (a)

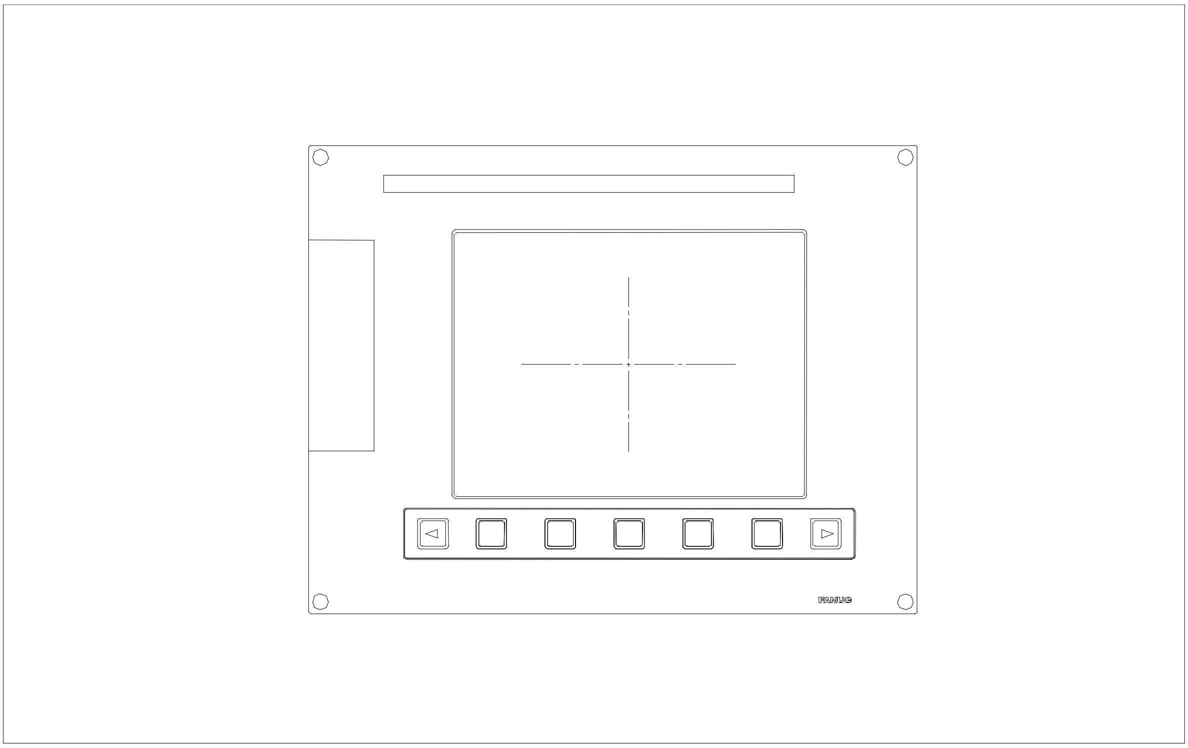
2 OPERATIONAL DEVICES

As operational devices, setting and display devices attached to the CNC, and machine operator's panels are available.
For machine operator's panels, refer to the relevant manual of the machine tool builder.

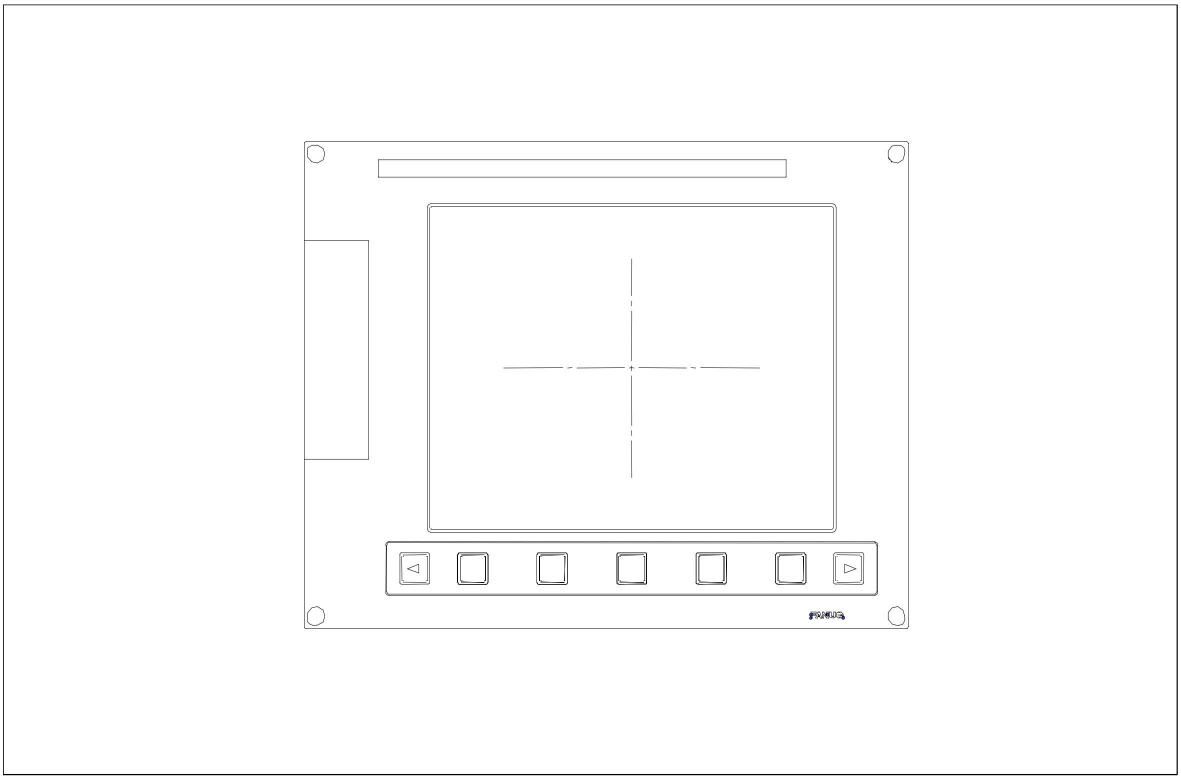
2.1 SETTING AND DEISPLAY UNITS

The setting and display units are shown in Subsections 2.1.1 to 2.1.5 of Part III.

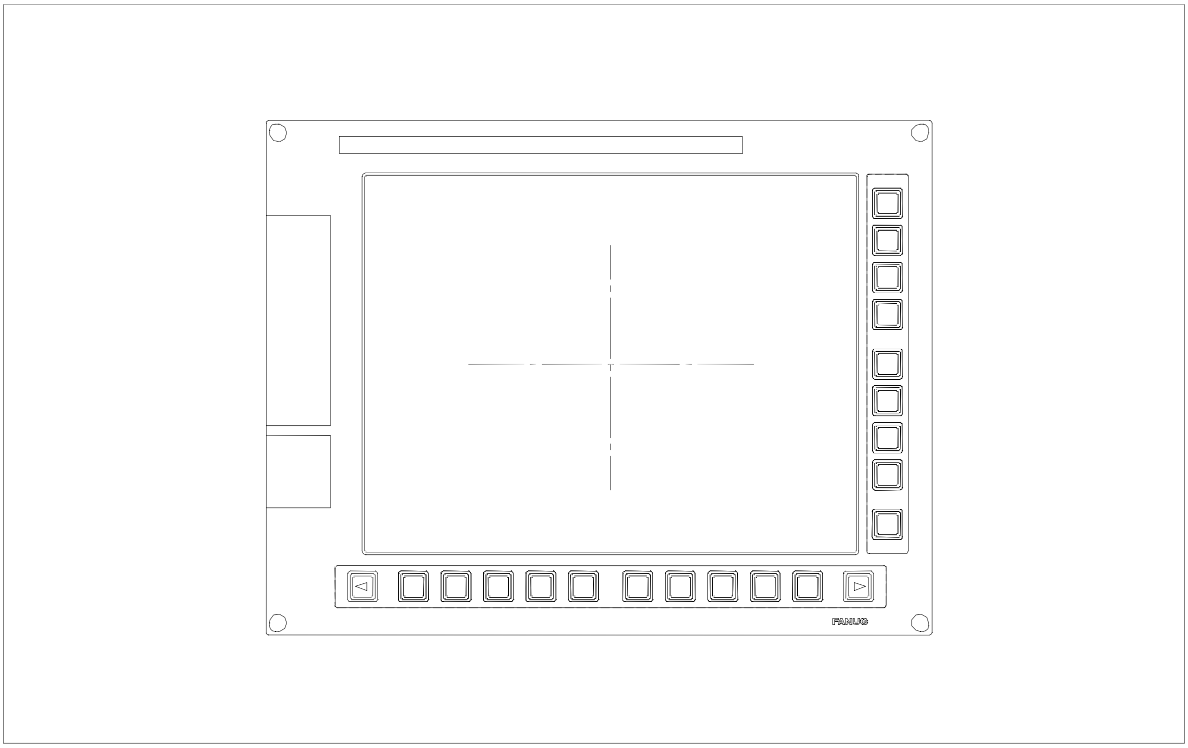
2.1.1 7.2" LCD CNC Display Panel



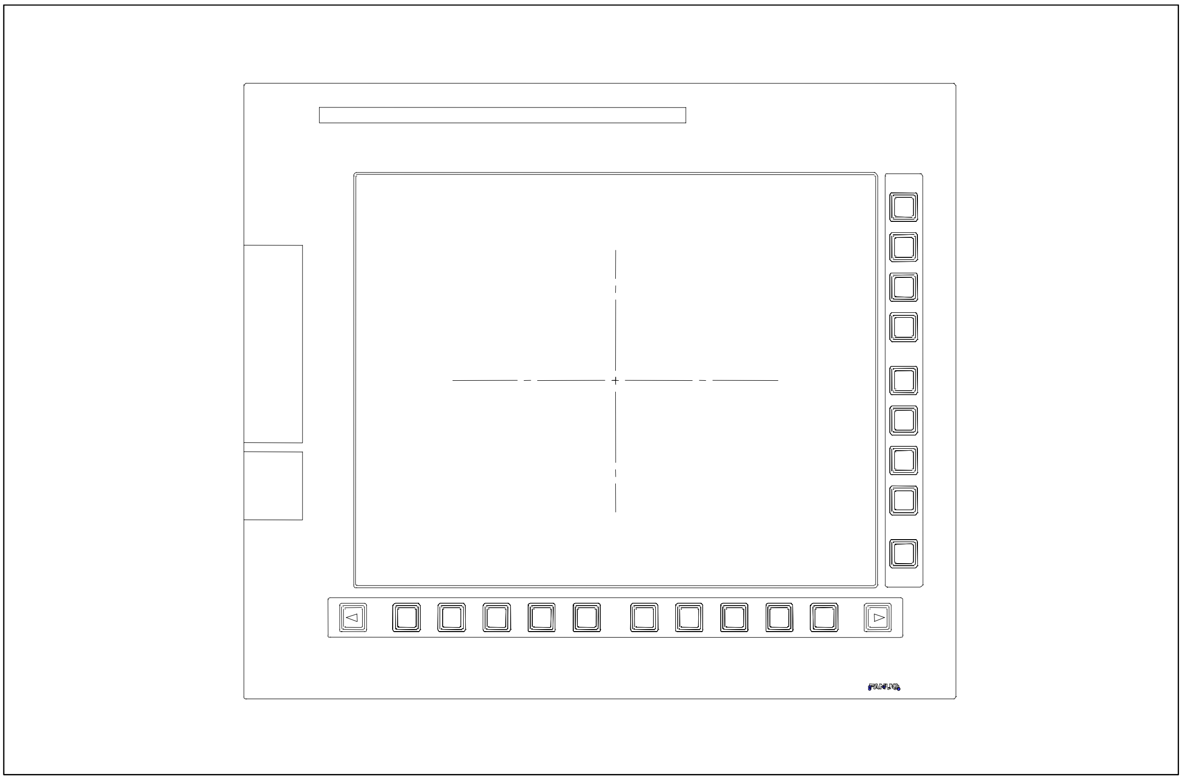
2.1.2 8.4" LCD CNC Display Panel



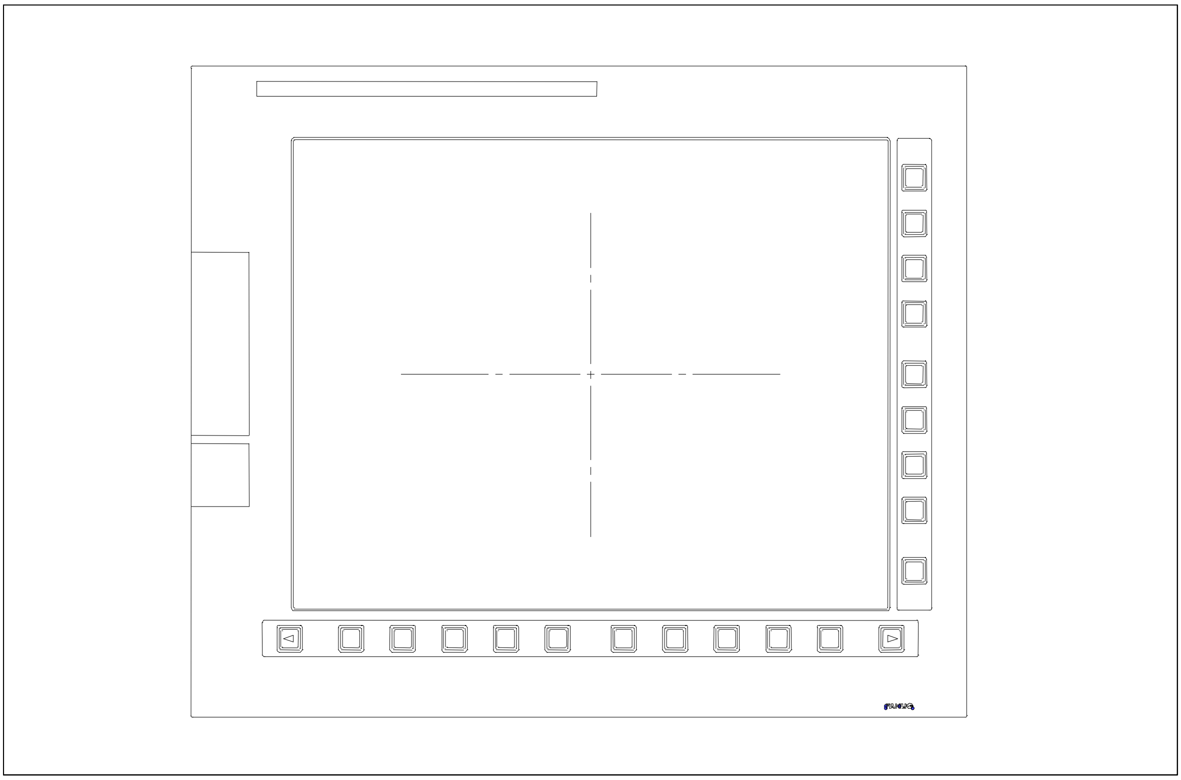
2.1.3 10.4" LCD CNC Display Panel



2.1.4 12.1" LCD CNC Display Panel

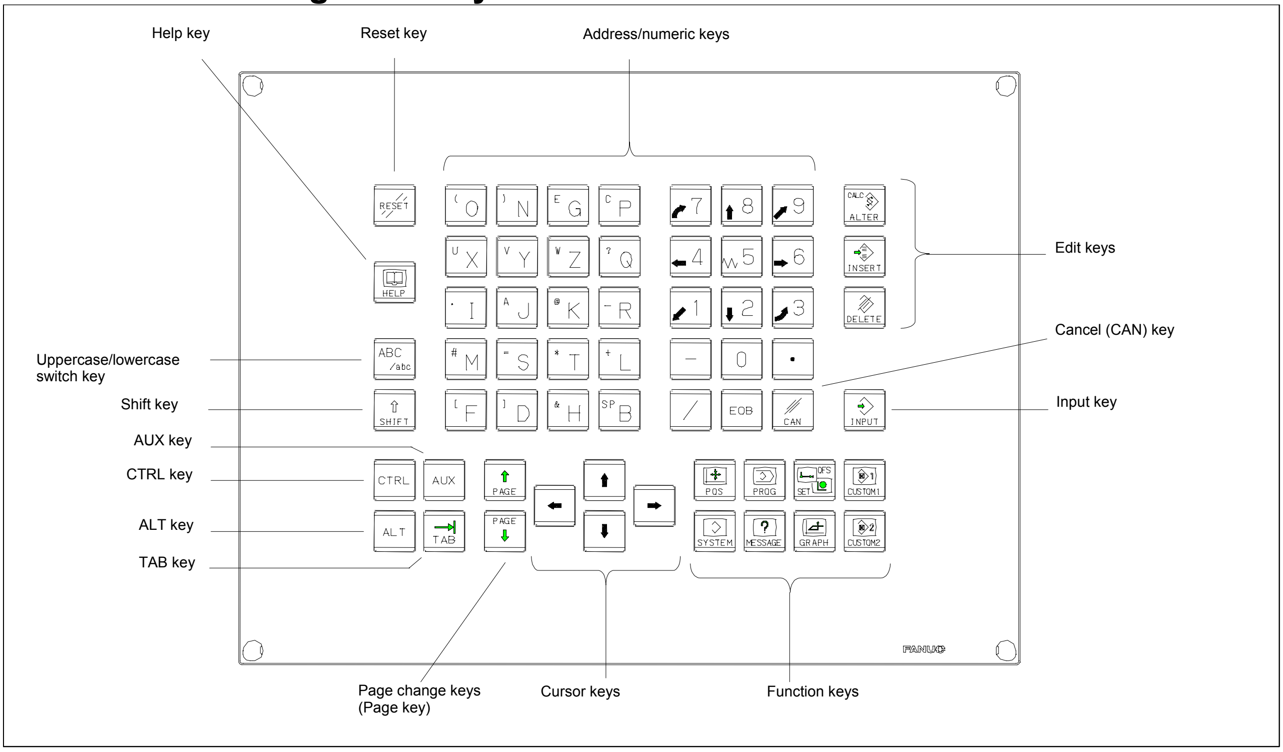


2.1.5 15" LCD CNC Display Panel

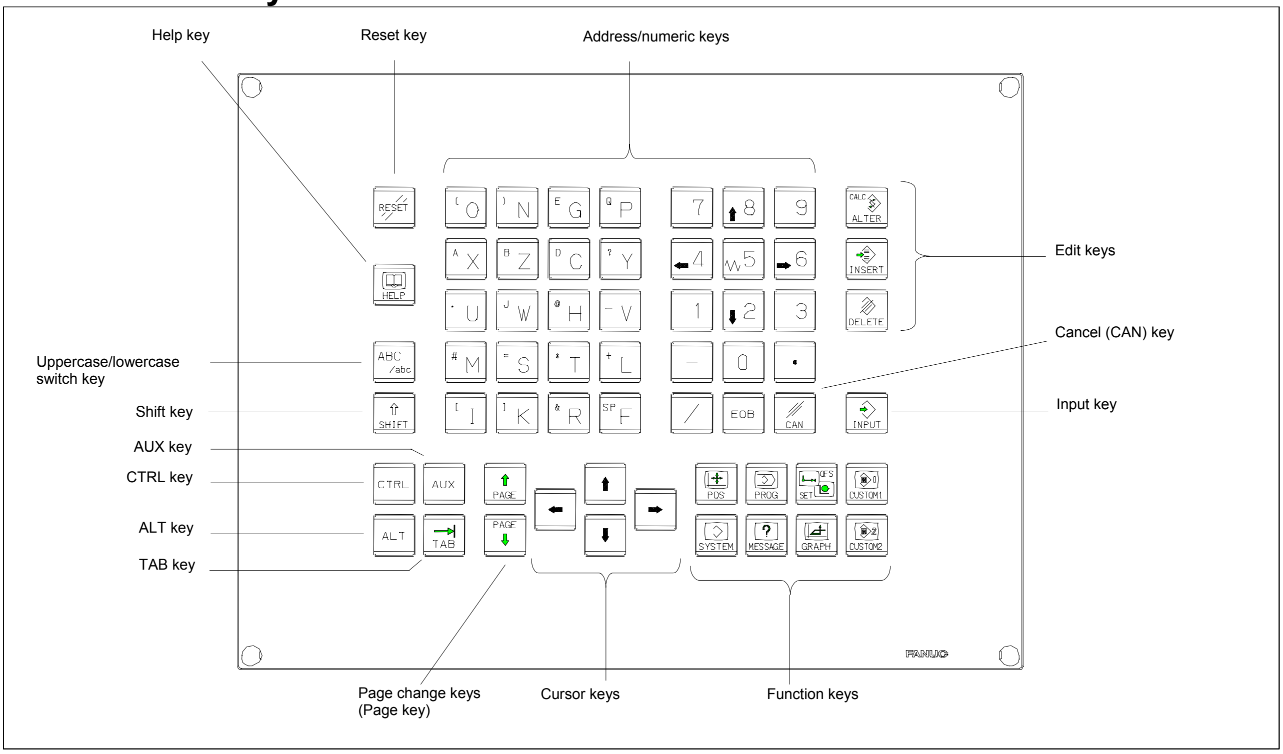


2.1.6 Standard MDI Unit (ONG Key)

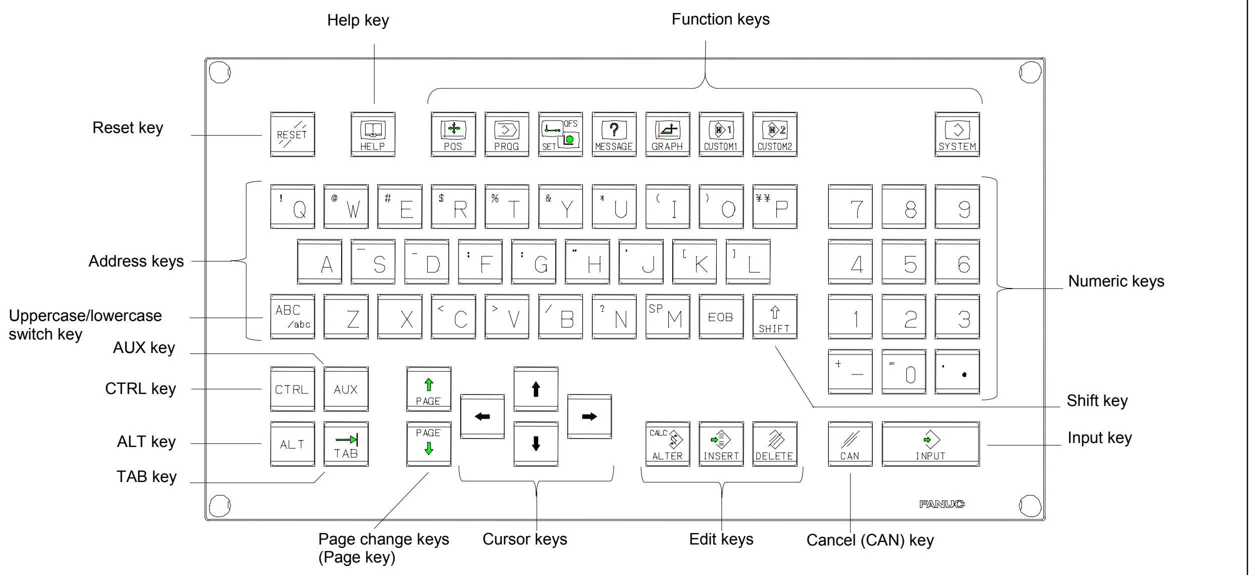
Unit with machining center system



Unit with lathe system

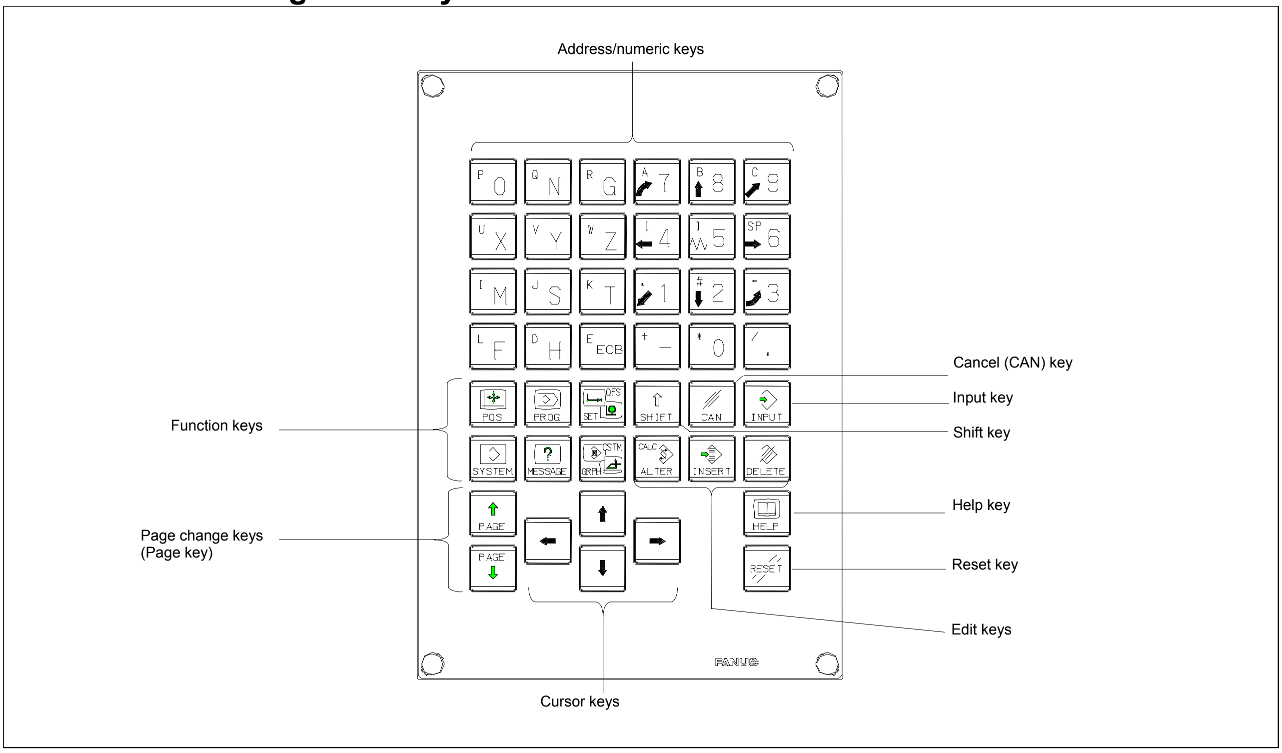


2.1.7 Standard MDI Unit (QWERTY Key)

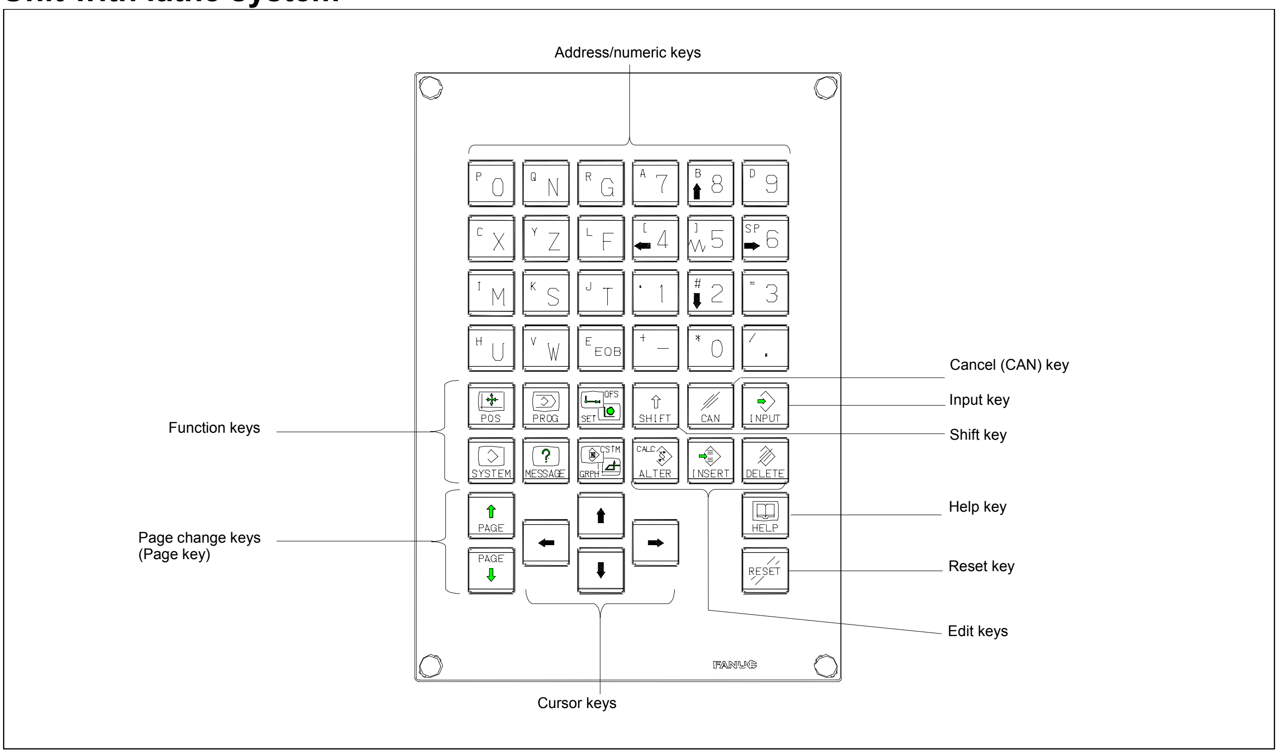


2.1.8 Small MDI Unit (ONG Key)

Unit with machining center system



Unit with lathe system



2.2 OPERATIONAL DEVICES

Table 2.2 (a) Explanation of the MDI keyboard

Number	Name	Explanation
1	RESET key RESET	Press this key to reset the CNC, to cancel an alarm, etc.
2	HELP key HELP	Press this button to use the help function when uncertain about the operation of an MDI key (help function).
3	Soft keys	The soft keys have various functions, according to the Applications. The soft key functions are displayed on the display unit.
4	Address and numeric keys N (4← . .	Press these keys to input alphabetic, numeric, and other characters.
5	SHIFT key SHIFT	Some keys have two characters on their keytop. Pressing the <SHIFT> key switches the characters. Special character # is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered.
6	INPUT key INPUT	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the screen. To copy the data in the key input buffer to the offset register, etc., press the <INPUT> key. This key is equivalent to the [INPUT] key of the soft keys, and either can be pressed to produce the same result.
7	CANCEL (CAN) key CAN	Press this key to delete the last character or symbol input to the key input buffer. Example) When the key input buffer displays > N001X100Z_ >N001X100Z_ and the cancel key CAN is pressed, Z is canceled and >N001X100_ is displayed.
8	Edit keys ALTER INSERT DELETE	Press these keys when editing the program. ALTER : ALTER INSERT : INSERT DELETE : DELETE
9	Function keys POS PROG . .	Press theses keys to switch display screens for each function. See III-2.3 for details of the function keys.
10	Cursor keys ← ↑ ↓ →	There are four different cursor move keys. → : This key is used to move the cursor to the right or in the forward direction. The cursor is moved in short units in the forward direction. ← : This key is used to move the cursor to the left or in the reverse direction. The cursor is moved in short units in the reverse direction. ↓ : This key is used to move the cursor in a downward or forward direction. The cursor is moved in large units in the forward direction. ↑ : This key is used to move the cursor in an upward or reverse direction. The cursor is moved in large units in the reverse direction.

Table 2.2 (a) Explanation of the MDI keyboard

Number	Name	Explanation
11	Page change keys (Page keys) ↑ PAGE PAGE ↓	Two kinds of page change keys are described below. PAGE ↓ : This key is used to changeover the page on the screen in the forward direction. ↑ PAGE : This key is used to changeover the page on the screen in the reverse direction.
12	Uppercase/lowercase switch key ABC /abc	Press this key to switch between uppercase and lowercase when entering alphabetic characters.
13	PC function key CTRL AUX ALT TAB	These keys are used with the personal computer function of the 300*i*/300*i*s, 310*i*/310*i*s, 320*i*/320*i*s.

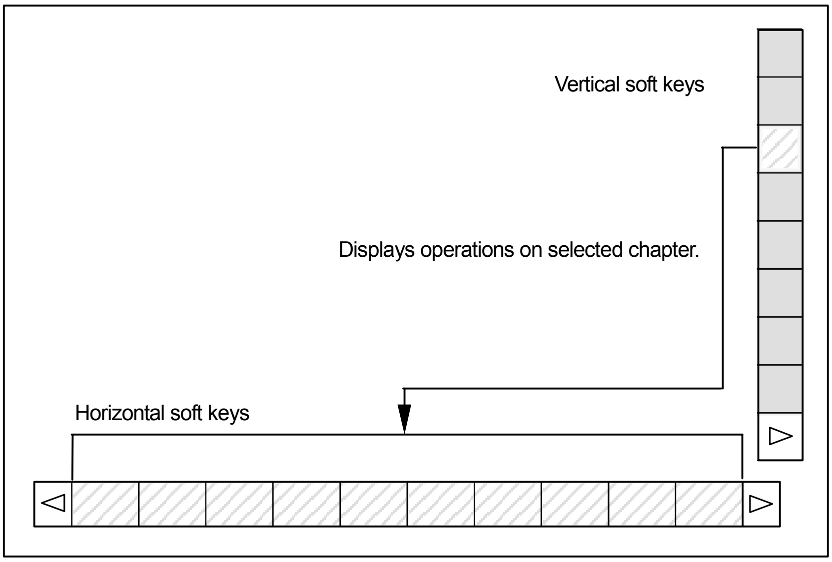
Explanation

- 15" LCD/MDI soft key configuration

The 15" LCD panel has 12 soft keys horizontally and 9 soft keys vertically.

As shown below, the 8 vertical soft keys and the lowermost key are used as chapter selection soft keys. By pressing each of these keys, the screen (chapter) belonging to each function can be selected. The horizontal 12 soft keys are used to perform operations on the screen selected with a vertical soft key.

For an LCD display with a touch panel, touch a soft key on the screen to select it.



In this manual, the descriptions below assume a 10.4" LCD display panel with 12 soft keys.

- Key operation with multi-path control

In the multi-path control, be sure to select the tool post for which data is specified, using the path selection switch on the machine operator's panel. Then, perform keyboard operation, such as displaying or specifying various data items, and editing a program.

2.3 FUNCTION KEYS AND SOFT KEYS

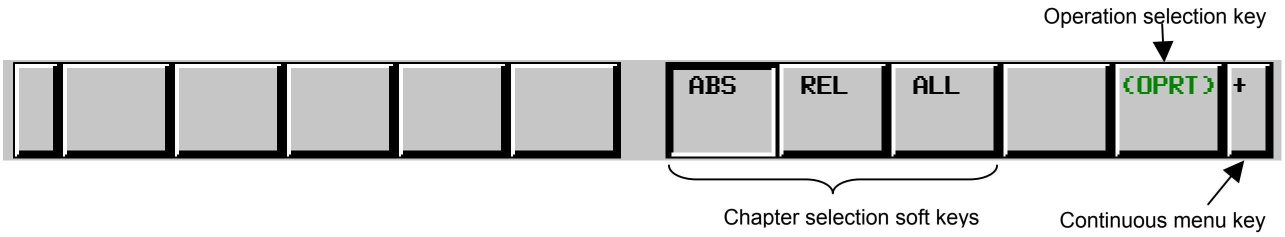
The function keys are used to select the type of screen (function) to be displayed. When a soft key (section select soft key) is pressed immediately after a function key, the screen (section) corresponding to the selected function can be selected.

2.3.1 General Screen Operations

- Procedure

1 By pressing a function key on the MDI panel, the chapter selection soft keys that belong to the function are displayed.

Example 1)

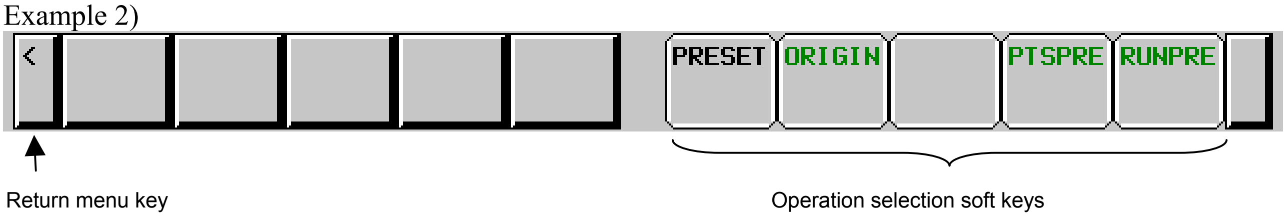


2 When one of the chapter selection soft keys is pressed, the screen of the chapter is displayed. If the soft key of a desired chapter is not displayed, press the continuous menu key.
In a chapter, a further choice may be made from multiple chapters.

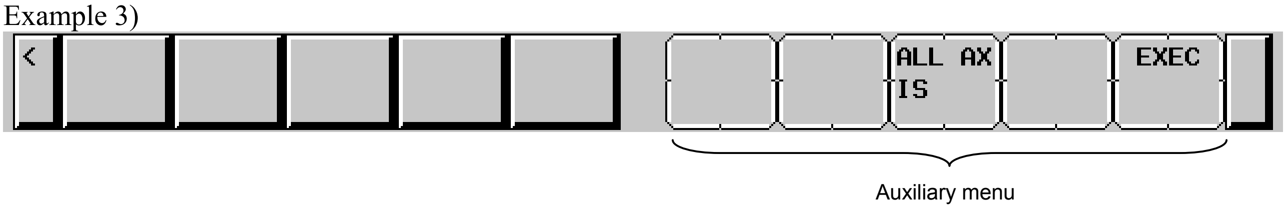
3 When the screen of a desired chapter is displayed, press the operation selection key to display operations to be performed.

4 Select a desired operation with the operation selection soft key. Depending on the operation to be executed, an auxiliary menu of soft keys is displayed. Perform an operation according to the indications on the auxiliary menu.

Example 2)



Example 3)



5 To return to the display of chapter selection soft keys, press the return menu key.

A general screen display procedure is provided above.
The actual display procedure varies from one screen to another.
For details, see each description of operation.

- Button design change depending on soft key state

The soft keys assume one of the following states, depending on the selection target:

- Chapter selection soft keys
- Operation selection soft keys
- Auxiliary menu of operation selection soft keys

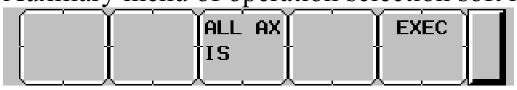
Depending on the state, the button images of the soft keys change. From the button images, which state the soft keys are assuming can be known.

Example)

- Chapter selection soft keys

- Operation selection soft keys

- Auxiliary menu of operation selection soft keys



2.3.2 Function Keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the MDI panel:

POS	Press this key to display the position screen.
PROG	Press this key to display the program screen.
OFFSET SETTING	Press this key to display the offset/setting screen.
SYSTEM	Press this key to display the system screen.
MESSAGE	Press this key to display the message screen.
GRAPH	Press this key to display the graphics screen.
CUSTOM1	Press this key to display the custom screen (conversational macro screen).
CUSTOM2	Unused.

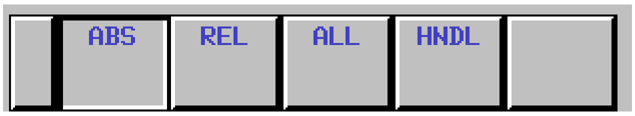
2.3.3 Soft Keys

By pressing a soft key after a function key, the corresponding screen of the function can be displayed.

The chapter selection soft keys of each function are described below. The horizontal four keys on the right-hand side are assigned to chapter selection soft keys. When multiple pages are used for chapter selection soft keys, [+] is displayed on the continuous menu key (rightmost soft key). Press the continuous menu key to switch between chapter selection soft keys.

NOTE
1. Press function keys to switch between screens that are used frequently.
2. Some soft keys are not displayed depending on the option configuration.

If position indications are provided on the left half of the screen when a key other than the function key POS is pressed, the left half of the soft keys is displayed as follows at all times:



Position display screen

The chapter selection soft keys that belong to the function key POS and the function of each screen are described below.

	(1)	(2)	(3)	(4)	(5)	
Page 1	ABS	REL	ALL	HNDL	(OPRT)	+

	(6)	(7)	(8)	(9)	(10)	
Page 2	MONI	5AXMAN			(OPRT)	+

Table 2.3.3 (a) Position display screen

No.	Chapter menu	Description
(1)	ABS	Selects the absolute coordinate display screen.
(2)	REL	Selects the relative coordinate display screen.
(3)	ALL	Selects the overall coordinate display screen.
(4)	HNDL	Selects the operation screen for manual handle operation.
(6)	MONI	Selects the screen for displaying the servo axis load meter, serial spindle load meter, and speedometer.
(7)	5AXMAN	Displays a handle pulse interrupt amount in manual feed for 5-axis machining.

Program screen

The chapter selection soft keys that belong to the function key PROG and the function of each screen are described below.

Page 1

(1)	(2)	(3)	(4)	(5)	
PROGRAM	FOLDER	NEXT	CHECK	(OPRT)	+

Page 2

(6)	(7)	(8)	(9)	(10)	
RSTR	JOG			(OPRT)	+

Table 2.3.3 (b) Program

No.	Chapter menu	Description
(1)	PROGRAM	Selects the screen for displaying a list of part programs currently registered.
(2)	FOLDER	Selects the screen for displaying a list of part programs currently registered.
(3)	NEXT	Selects the screen for displaying the command values of the block currently executed and the next block to be executed among the command values.
(4)	CHECK	Selects the screen for displaying programs, position data, modal information, and so forth simultaneously.
(6)	RSTR	Selects the operation screen for restarting an interrupted program operation.
(7)	JOG	Selects the screen for executing, in the JOG mode, data specified in the program format from the MDI.

Offset/setting screen

The chapter selection soft keys that belong to the function key OFFSET SETTING and the function of each screen are described below.

	(1)	(2)	(3)	(4)	(5)	
Page 1	OFFSET	SETTING	WORK		(OPRT)	+
	(6)	(7)	(8)	(9)	(10)	
Page 2	MACRO		OPR	TOOL MANAGER	(OPRT)	+
	(11)	(12)	(13)	(14)	(15)	
Page 3	OFST.2	W.SHFT	GEOM.2		(OPRT)	+
	(16)	(17)	(18)	(19)	(20)	
Page 4		PR-LV			(OPRT)	+
	(21)	(22)	(23)	(24)	(25)	
Page 5	CHUCK TAIL	LANG.	PROTECT	GUARD	(OPRT)	+
	(26)	(27)	(28)	(29)	(30)	
Page 6				TOOL LIFE	(OPRT)	+
	(31)	(32)	(33)	(34)	(35)	
Page 7	WORK SET ER				(OPRT)	+

Table 2.3.3 (c) Offset

No.	Chapter menu	Description
(1)	OFFSET	Selects the screen for setting tool offset values.
(2)	SETTING	Selects the screen for setting the setting parameters.
(3)	WORK	Selects the screen for setting a workpiece coordinate system offset.
(6)	MACRO	Selects the screen for setting macro variables.
(8)	OPR	Selects the screen for operating some operation switches on the machine operator's panel as soft switches.
(9)	TOOL MANAGER	Selects the screen for setting data related to tool management.
(11)	OFST.2	Selects the screen for setting a Y-axis offset.
(12)	W.SHFT	Selects the screen for setting a workpiece coordinate system shift value.
(13)	GEOM.2	Selects the screen for setting a second geometry offset.
(17)	PR-LV	Selects the screen for setting a precision level.
(21)	CHUCK TAIL	Selects the chuck tail stock barrier screen.
(22)	LANG.	Selects the screen for setting a display language.
(23)	PROTECT	Selects the screen for setting data protection.
(24)	GUARD	Selects the screen for setting wrong operation prevention.
(29)	TOOL LIFE	Selects the screen for operations and setting related to tool life management.
(31)	WRK ERR COMP	Selects the screen for setting errors related to workpiece mounting position.

System screen

The chapter selection soft keys that belong to the function key SYSTEM and the function of each screen are described below.

	(1)	(2)	(3)	(4)	(5)	
Page 1	PARAM	DGNOS	SERVO GUIDEM	SYSTEM	(OPRT)	+
	(6)	(7)	(8)	(9)	(10)	
Page 2	MEMORY	PITCH	SERVO PARAM	SP.SET	(OPRT)	+
	(11)	(12)	(13)	(14)	(15)	
Page 3	PMC MAINTE	PMC LADDER	PMC CONFIG		(OPRT)	+
	(16)	(17)	(18)	(19)	(20)	
Page 4	MCNG TUNING	ALL IO	ALL IO	OPEHIS	(OPRT)	+
	(21)	(22)	(23)	(24)	(25)	
Page 5	COLOR	MAINTE	M-INFO	W. DGNS	(OPRT)	+
	(26)	(27)	(28)	(29)	(30)	
Page 6		FSSB	PRMTUN		(OPRT)	+
	(31)	(32)	(33)	(34)	(35)	
Page 7	EMBEDDED PORT	PCMCIA LAN	ETHNET BOARD	PROFI-B US MST	(OPRT)	+

	(36)	(37)	(38)	(39)	(40)	
Page 8	REMOTE DIAG	M CODE			(OPRT)	+

	(41)	(42)	(43)	(44)	(45)	
Page 9	PROFI SLAVE	DEVNET MASTER			(OPRT)	+

	(46)	(47)	(48)	(49)	(50)	
Page 10	DUAL CHECK	R.TIME MACRO			(OPRT)	+

Table 2.3.3 (d) System

No.	Chapter menu	Description
(1)	PARAM	Selects the screen for setting parameters.
(2)	DGNOS	Selects the screen for displaying CNC state.
(3)	SERVO GUIDEM	Selects the screen for displaying the servo guide mate.
(4)	SYSTEM	Selects the screen for displaying the current system status.
(6)	MEMORY	Selects the screen for displaying the contents of memory.
(7)	PITCH	Selects the screen for setting pith error compensation.
(8)	SERVO PARAM	Selects the screen for setting the servo-related parameters.
(9)	SP.SET	Selects the screen for spindle-related setting.
(11)	PMC MAINTE	Selects the screen related to PMC maintenance such as PMC signal state monitoring and tracing, and PMC parameter display/editing.
(12)	PMC LADDER	Selects the screen related to ladder display/editing.
(13)	PMC CONFIG	Displays the screen for displaying/editing data other than ladders that makes up a sequence program and for setting the PMC function.
(16)	MCNG TUNING	Displays the screen for setting the parameter set for emphasis on speed (LV1) or emphasis on precision (LV10).
(17)	ALL IO	Selects the screen for data I/O.
(18)	ALL IO	Selects the screen for data input to and output from the memory card.
(19)	OPEHIS	Selects the screen for displaying the history of operations performed by the operator and issued alarms.
(21)	COLOR	Selects the screen for setting colors to be used on the screen.
(22)	MAINTE	Selects the screen for setting maintenance items to be managed periodically.
(23)	M-INFO	Selects the screen for displaying information about maintenance performed.

No.	Chapter menu	Description
(24)	W.DGNS	Selects the screen for displaying data such as servo positional deviation values, torque values, machine signals, and so forth as graphs.
(27)	FSSB	Selects the screen for making settings related to the high-speed serial servo bus (FSSB: Fanuc Serial Servo Bus).
(28)	PRMTUN	Selects the screen for setting parameters necessary for start-up and tuning.
(31)	EMBEDDED PORT	Selects the screen for making settings related to the embedded Ethernet (embedded port).
(32)	PCMCIA LAN	Selects the screen for making settings related to the embedded Ethernet (PCMCIA Ethernet card).
(33)	ETHNET BOARD	Selects the screen for making settings related to the fast Ethernet/fast data server.
(34)	PROFI-BUS MST	Selects the screen for making settings related to the profibus master function.
(37)	M CODE	Selects the screen for setting an M code group.
(41)	PROFI SLAVE	Selects the screen for making settings related to the profibus slave function.
(42)	DEVNETMASTER	Selects the screen for making settings related to the DeviceNet master function.
(46)	DUAL CHECK	Selects the screen for making settings related to the dual check safety function.
(47)	R.TIMEMACRO	Selects the screen for making settings related to the real-time custom macro function.

Message screen

The chapter selection soft keys that belong to the function key MESSAGE and the function of each screen are described below.

	(1)	(2)	(3)	(4)	(5)	
Page 1	ALARM	MSG	HISTRY	MSGHIS	(OPRT)	+

	(6)	(7)	(8)	(9)	(10)	
Page 2	BUILT-IN LOG	PCMCIA LOG	BOARD LOG		(OPRT)	+

Table 2.3.3 (e) Message

No.	Chapter menu	Description
(1)	ALARM	Selects the alarm message screen.
(2)	MSG	Selects the operator message screen.
(3)	HISTRY	Selects the screen for displaying the details of alarms issued so far.
(4)	MSGHIS	Selects the external operator message screen.
(6)	BUILT-IN LOG	Selects the screen for displaying error messages related to the embedded Ethernet (embedded port).
(7)	PCMCIA LOG	Selects the screen for displaying error messages related to the embedded Ethernet (PCMCIA Ethernet card).
(8)	BOARD LOG	Selects the screen for displaying error messages related to the fast Ethernet/fast data server.

Graphic screen

The chapter selection soft keys that belong to the function key GRAPH and the function of each screen are described below.

	(1)	(2)	(3)	(4)	(5)	
Page 1	PARAM	GRAPH			(OPRT)	+

Table 2.3.3 (f) Graphic

No.	Chapter menu	Description
(1)	PARAM	Selects the screen for setting graphic parameters.
(2)	GRAPH	Selects the screen for graphically displaying the tool path.

2.3.4 Key Input and Input Buffer

When an address and a numeric key are pressed, the character corresponding to that key is input once into the key input buffer. The contents of the key input buffer is displayed at the bottom of the LCD screen.
In order to indicate that it is key input data, a ">" symbol is displayed immediately in front of it. A "_" is displayed at the end of the key input data indicating the input position of the next character.

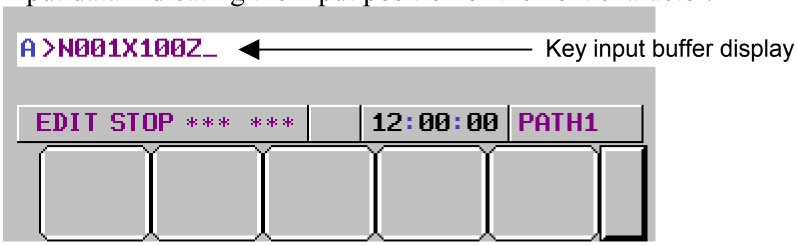


Fig. 2.3.4 (a) Key input buffer display

- Switching between upper and lower key characters

To input the lower character of the keys that have two characters inscribed on them, first press the SHIFT key and then the key in question.
When the SHIFT key is pressed, "_" indicating the next character input position changes to "^". Now lowercase characters can be entered (shift state).
When a character is input in shift status the shift status is canceled. Furthermore, if the SHIFT key is pressed in shift status, the shift status is canceled.
It is possible to input up to 32 characters at a time in the key input buffer.
Press the CAN key to cancel a character or symbol input in the key input buffer.

(Example)
When the key input buffer displays
>N001X100Z_
and the cancel CAN key is pressed, Z is canceled and
>N001X100_
is displayed.

- Switching between uppercase and lowercase alphabetic characters

When entering alphabetic characters, the user can switch between uppercase and lowercase.

By pressing the uppercase/lowercase switch key ABC/abc, the display of the key input buffer changes to enable uppercase or lowercase alphabetic characters to be entered as described below.

Example)

A> A B C _
State enabling uppercase input

a> a b c _
State enabling lowercase input

2.3.5 Warning Messages

After a character or number has been input from the MDI panel, a data check is executed when INPUT key or a soft key is pressed. In the case of incorrect input data or the wrong operation a flashing warning message will be displayed on the status display line.

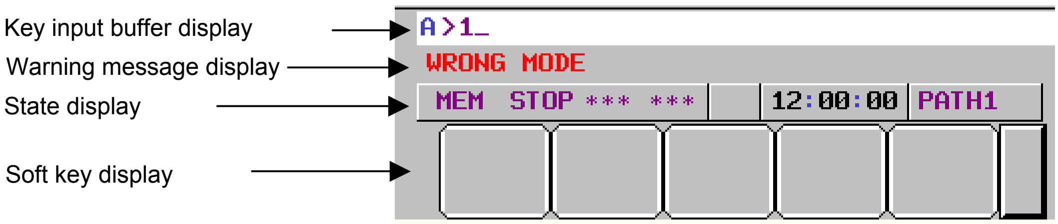


Fig. 2.3.5 (a) Warning message display

Table 2.3.5 (a) Warning Messages

Warning message	Content
FORMAT ERROR	The format is incorrect.
WRITE PROTECT	Key input is invalid because of data protect key or the parameter is not write enabled.
DATA IS OUT OF RANGE	The input value exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.
WRONG MODE	Parameter input is not possible in any mode other than MDI mode.
EDIT REJECTED	It is not possible to edit in the current CNC status.

2.4 EXTERNAL I/O DEVICES

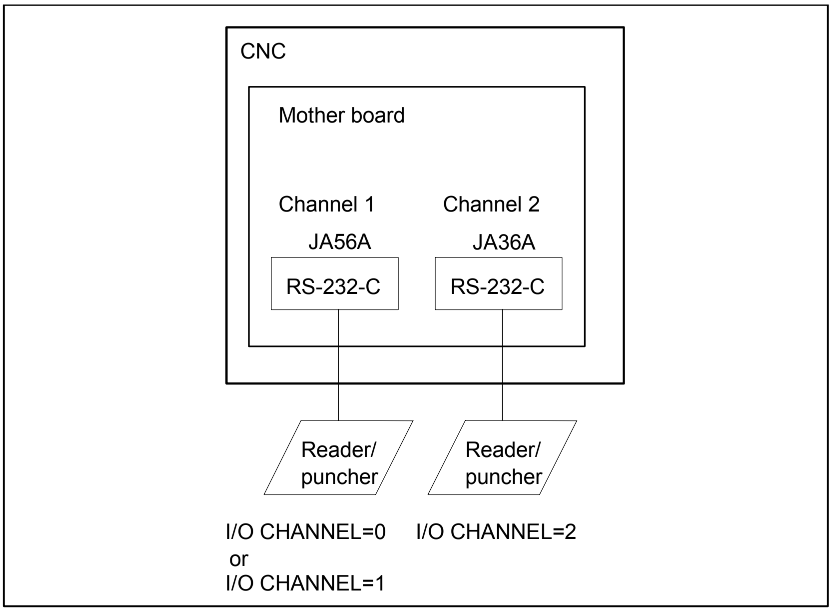
External I/O devices such as a memory card are available.
By using an external I/O device such as a memory card, the following data can be input or output:

1. Programs
2. Offset data
3. Parameters
4. Custom macro common variables

For how to input or output data and input data from or output it to a memory card, see III-8.

- Parameter setting

Before an external input/output device can be used, parameters must be set as follows.



This CNC has a total of two channels of reader/puncher interfaces. It also has a memory card interface. The input/output device to be used is specified by setting the channel (interface) connected to that device in setting parameter I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance. (These settings are not required for the memory card interface.)
For channel 1, two combinations of parameters to specify the input/output device data are provided.
The following shows the interrelation between the reader/puncher interface parameters for the channels.

$\boxed{0020}$ I/O CHANNEL or foreground input	Input/output channel number (parameter No.0020) ↓
Set channels to be used for data input/output.	
I/O CHANNEL (0 to 5) =0 : Channel 1 =1 : Channel 1 =2 : Channel 2 =3 : Channel 3 : : :	I/O CHANNEL=0 (Channel 1): $\boxed{0101}$ Stop bit and other data; $\boxed{0102}$ Number specified for the input/output device; $\boxed{0103}$ Baud rate I/O CHANNEL=1 (Channel 1): $\boxed{0111}$ Stop bit and other data; $\boxed{0112}$ Number specified for the input/output device; $\boxed{0113}$ Baud rate
Input/output to and from the memory card interface, etc. is also possible.	I/O CHANNEL=2 (Channel 2): $\boxed{0121}$ Stop bit and other data; $\boxed{0122}$ Number specified for the input/output device; $\boxed{0123}$ Baud rate
When IO4 is set $\boxed{0021}$ Foreground output $\boxed{0022}$ Background input $\boxed{0023}$ Background input	: : : I/O CHANNEL=5
The channel setting is the same as No.0020.	

2.5 POWER ON/OFF

2.5.1 Turning on the Power

Procedure of turning on the power

Procedure

1. Check that the appearance of the CNC machine tool is normal. (For example, check that front door and rear door are closed.)
2. Turn on the power according to the manual issued by the machine tool builder.
3. After the power is turned on, check that the position screen is displayed. An alarm screen is displayed if an alarm occurs upon power-on.

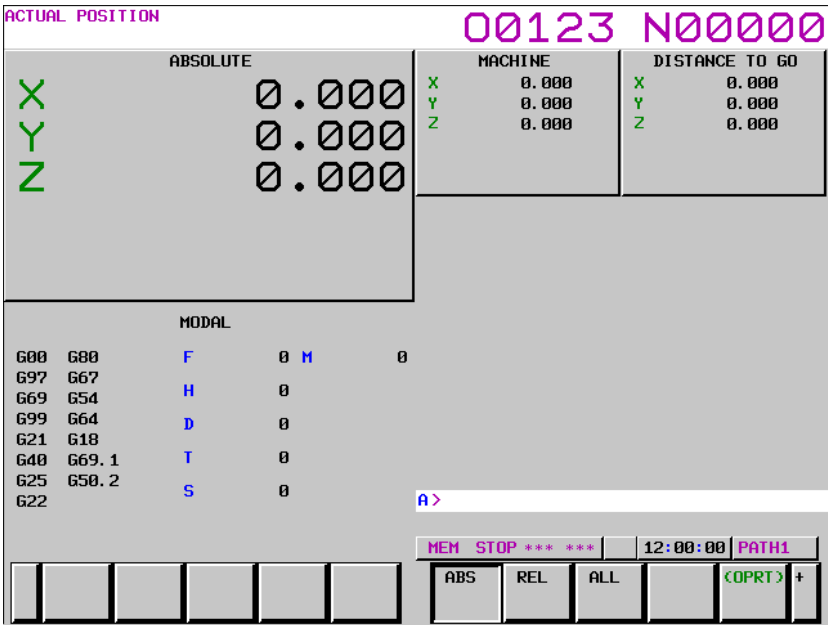


Fig. 2.5.1 (a) Position screen (for machining center system)

4. Check that the fan motor is rotating.

> ⚠ **WARNING**
> Until the positional or alarm screen is displayed at the power on, do not touch them. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

2.5.2 Power Disconnection

Procedure of power disconnection

Procedure

1. Check that the LED indicating the cycle start is off on the operator's panel.
2. Check that all movable parts of the CNC machine tool is stopping.
3. If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.
4. Continue to press the <POWER OFF> button for about 5 seconds.
5. Refer to the machine tool builder's manual for turning off the power to the machine.

3 MANUAL OPERATION

MANUAL OPERATION are six kinds as follows :

3.1 MANUAL REFERENCE POSITION RETURN
3.2 JOG FEED (JOG)
3.3 INCREMENTAL FEED
3.4 MANUAL HANDLE FEED
3.5 MANUAL ABSOLUTE ON AND OFF
3.6 RIGID TAPPING BY MANUAL HANDLE
3.7 MANUAL NUMERICAL COMMAND
3.8 MANUAL FEED FOR 5-AXIS MACHINING
3.9 DISTANCE CODED LINEAR SCALE INTERFACE
3.10 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)

3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :
The tool is moved in the direction specified in parameter ZMI (No. 1006#5) for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 1424,1421, and 1425).
Four step rapid traverse override is effective during rapid traverse.
When the tool has returned to the reference position, the reference position return completion LED goes on. The tool generally moves along only a single axis, but can move along three axes simultaneously when specified so in parameter JAX (No.1002#0).

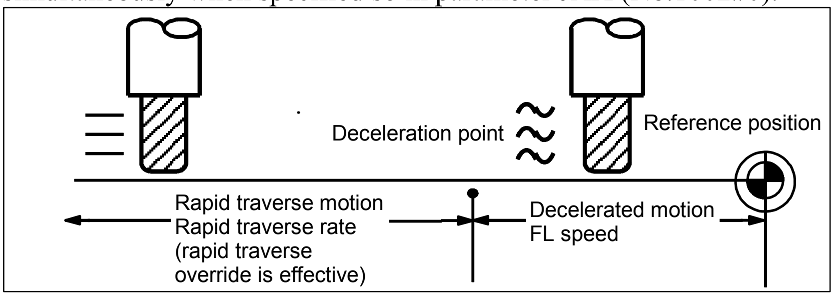


Fig. 3.1 (a) Manual reference position return

Procedure for manual reference position return

Procedure

1 Press the reference position return switch, one of the mode selection switches.
2 To decrease the feedrate, press a rapid traverse override switch. When the tool has returned to the reference position, the reference position return completion LED goes on.
3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool can be moved along three axes simultaneously when specified so in an appropriate parameter setting. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter.
4 Perform the same operations for other axes, if necessary.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

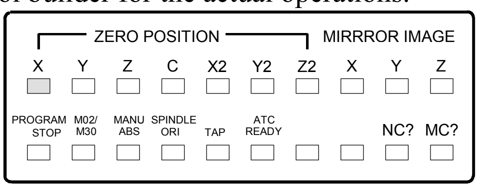


Fig. 3.1 (b)

Explanation

- Automatically setting the coordinate system

Parameter ZPR (No. 1201#0) is used for automatically setting the coordinate system. When ZPR is set, the coordinate system is automatically determined when manual reference position return is performed.

When α, β and γ are set in parameter 1250, the workpiece coordinate system is determined so that reference point on the tool holder or the position of the tip of the reference tool is X= α, Y = β, Z = γ when reference position return is performed. This has the same effect as specifying the following command for reference position return:

G92X$\underline{\alpha}$Y$\underline{\beta}$Z$\underline{\gamma}$;

However, when options of the workpiece coordinate system is selected, it is not able to use.

Limitation

- Moving the tool again

Once the reference position return completion LED lights at the completion of reference position return, the tool does not move unless the reference position return switch is turned off.

- Reference position return completion LED

The reference position return completion LED is extinguished by either of the following operations:

- Moving from the reference position.
- Entering an emergency stop state.

- The distance to return to reference position

For the distance (Not in the deceleration condition) to return the tool to the reference position, refer to the manual issued by the machine tool builder.

3.2 JOG FEED (JOG)

In the jog mode, pressing a feed axis and direction selection switch on the
machine operator's panel continuously moves the tool along the selected axis in the selected direction.
The jog feedrate is specified in a parameter (No.1423).
The jog feedrate can be adjusted with the jog feedrate override dial.
Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate (No. 1424) regardless of the position of the jog feedrate override dial. This function is called the manual rapid traverse.
Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter JAX (No.1002#0).

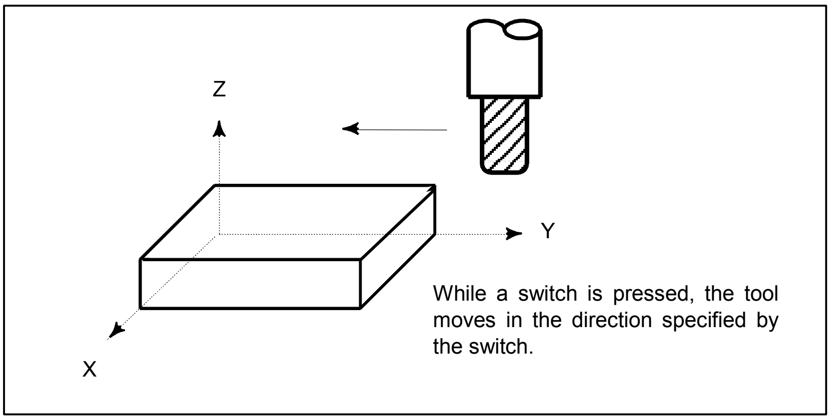


Fig. 3.2 (a) Jog Feed (JOG)

Procedure for JOG feed

Procedure

1 Press the jog switch, one of the mode selection switches.
2 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in a parameter (No. 1423). The tool stops when the switch is released.
3 The jog feedrate can be adjusted with the jog feedrate override dial.
4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate while the rapid traverse switch is pressed. Rapid traverse override by the rapid traverse override switches is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Manual per revolution feed

The manual per revolution feed is enabled for jog feed by setting parameter JRV (No. 1402 #4).
During the manual per revolution feed, the tool is jogged at the feedrate that is obtained by multiplying the spindle speed by the feed amount per revolution, which is calculated by multiplying the jog feedrate override value by the feed amount per revolution specified with the parameter (No. 1423).
During manual per revolution feed, the tool is jogged at the following feedrate:
Feed distance per rotation of the spindle (mm/rev) (specified with parameter No. 1423) × JOG feedrate override × actual spindle speed (rev/min).

Limitation

- Acceleration/deceleration for rapid traverse

Feedrate, time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.

- Change of modes

Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

- Rapid traverse prior to reference position return

If reference position return is not performed after power-on, pushing rapid traverse button does not actuate the rapid traverse but the remains at the JOG feedrate. This function can be disabled by setting parameter RPD (No.1401#01).

3.3 INCREMENTAL FEED

In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.
With using bit 2 (HNT) of parameter No. 7103, each step can be additionally 10 times the lest input increment.
The feedrate set in parameter No. 1423 is applied.
By using the manual feedrate override signal, the feedrate can be increased or decreased.
The tool can also be moved at the rapid traverse rate by using the manual rapid traverse selection signal, independent of the manual feedrate override signal.

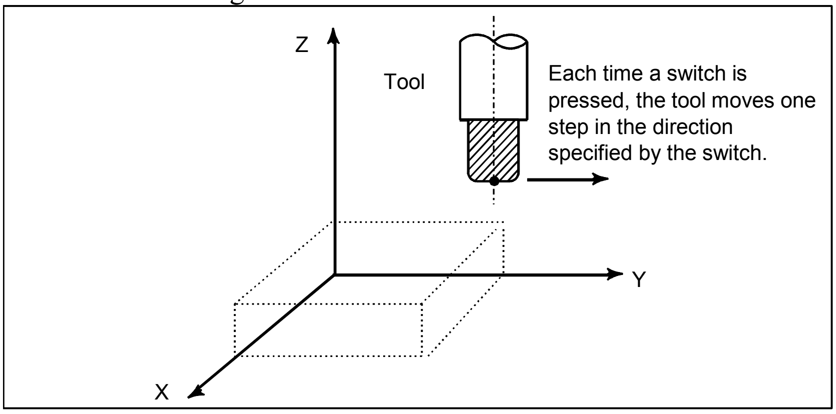


Fig. 3.3 (a) Incremental feed

Procedure for incremental feed

Procedure

1 Press the INC switch, one of the mode selection switches.
2 Select the distance to be moved for each step with the magnification dial.
3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.
4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate.
Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Travel distance specified with a diameter

T

The distance the tool travels along the X-axis can be specified with a diameter.

3.4 MANUAL HANDLE FEED

In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. One of four types of magnifiers selected with MP1 and MP2 <G019#4 and #5> can be applied. With bit 2 (HNT) of parameter No. 7103, the minimum distance can be further 10 times greater.
The number of manual pulse generators depends on the option as follows:

- Control on 1 manual handle: Up to 1
- Control on 2 or 3 manual handles:
  Up to 3 (Up to three manual pulse generators can be moved at a time.)

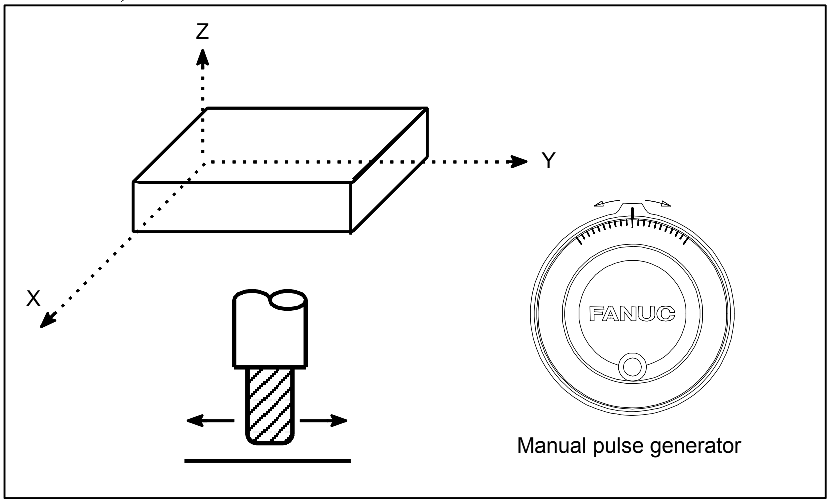


Fig. 3.4 (b) Manual handle feed

Procedure for manual handle feed

Procedure

1. Press the handle switch, one of the mode selection switches.
2. Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.
3. Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.
4. Move the tool along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduations.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Availability of manual pulse generator in Jog mode (JHD)

Parameter JHD (No. 7100#0) enables or disables the manual handle feed in the JOG mode.
When the parameter JHD(No. 7100#0) is set 1,both manual handle feed and incremental feed are enabled.

- Manual handle feed in TEACH IN JOG mode (THD)

By setting bit 1 (THD) of parameter No. 7100, manual handle feed in TEACH IN JOG mode can be enabled or disabled.

- A command to the MPG exceeding rapid traverse rate (HPF)

Parameter No. 7117 enables the following switching:

Set value 0 :

The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored. (The distance the tool is moved may not match the graduations on the manual pulse generator.)

Other than 0 :

The feedrate is clamped to the rapid traverse rate. However, those handle pulses beyond the rapid traverse rate are not ignored, but are processed as follows in connection with the manual handle feed travel distance selection signals:
(No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

When the magnification selected by manual handle feed travel distance selection signals is m and parameter No. 7117 is n:

When $n < m$: The feedrate is clamped to the value of parameter No. 7117.

When $n \geq m$: The feedrate is clamped to an integral multiple of a selected magnifier. If the mode is changed, however, a stop may not occur at an integral-multiple position.

- Upper feedrate limit in manual handle feed

The upper feedrate limit depends on the input signal (maximum manual handle feedrate switch signal HNDLF) from the PMC as follows:

- When HNDLF is set to 0, the feedrate is clamped to the manual rapid traverse rate (parameter No. 1424).
- When HNDLF is set to 1, the feedrate is clamped to the feedrate set in parameter No. 1434.

- Movement direction of an axis to the rotation of MPG (HNGX)

Parameter HNGx (No. 7102#0) switches the direction of MPG in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

This parameter is valid only for the following functions:

- Manual handle feed
- Manual handle interruption

Limitation

> ⚠ **WARNING**
> Rotating the handle quickly with a large magnification such as ×100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

> **NOTE**
> Rotate the manual pulse generator at a rate of five rotations per second or lower. If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.

3.5 MANUAL ABSOLUTE ON AND OFF

Whether the distance the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operator's panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is not added to the coordinates.

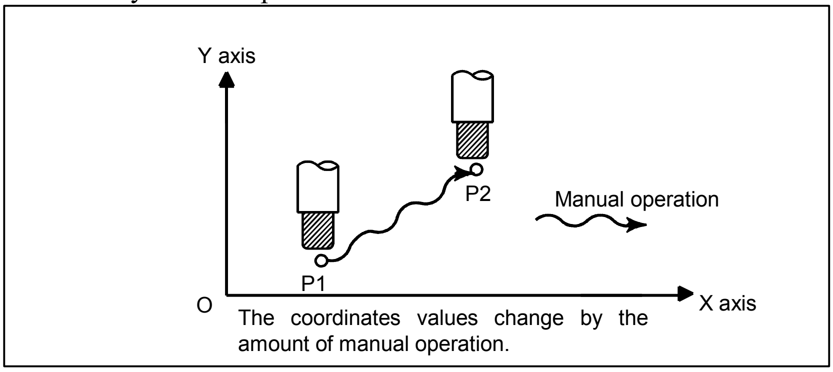


Fig. 3.5 (a) Coordinates with the switch ON

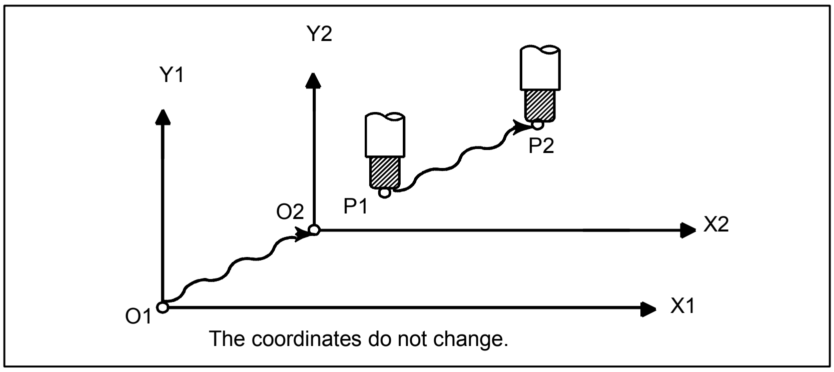


Fig. 3.5 (b) Coordinates with the switch OFF

Explanation

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

```
G01G90  X100.0Y100.0F010 ;          <1>
        X200.0Y150.0      ;          <2>
        X300.0Y200.0      ;          <3>
```

Fig. 3.5 (c) Program example

The subsequent figures use the following notation:

⟶ Movement of the tool when the switch is on

- - -➤ Movement of the tool when the switch is off

The coordinates after manual operation include the distance the tool is moved by the manual operation. When the switch is off, therefore, subtract the distance the tool is moved by the manual operation.

- Manual operation after the end of block

Coordinates when block <1> has been executed after manual operation (X-axis +20.0, Y-axis +100.0) at the end of movement of block <2>.

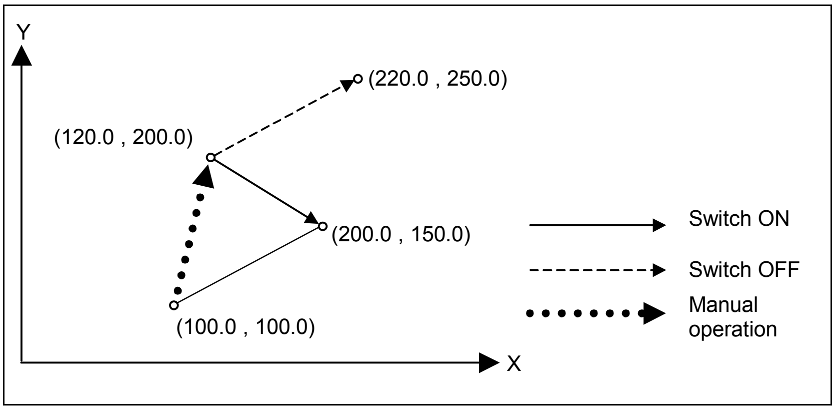


Fig. 3.5 (d) Manual operation after the end of block

- Manual operation after a feed hold

Coordinates when the feed hold button is pressed while block <2> is being executed, manual operation (Y-axis + 75.0) is performed, and the cycle start button is pressed and released.

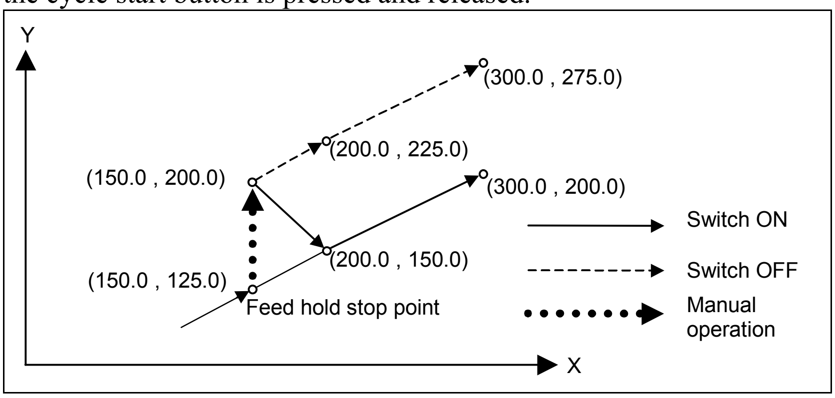


Fig. 3.5 (e) Manual operation after a feed hold

- When reset after a manual operation following a feed hold

Coordinates when the feed hold button is pressed while block <2> is being executed, manual operation (Y-axis +75.0) is performed, the control unit is reset with the RESET button, and block <2> is read again

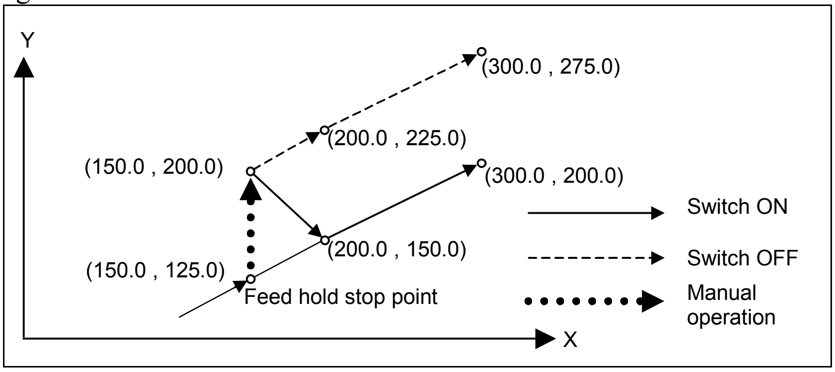


Fig. 3.5 (f) When reset after a manual operation following a feed hold

- When a movement command in the next block is only one axis

When there is only one axis in the following command, only the commanded axis returns.

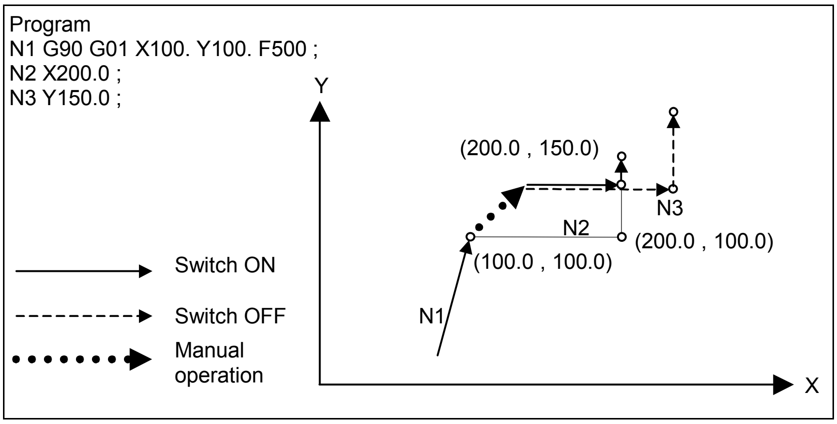


Fig. 3.5 (g) When a movement command in the next block is only one axis

- When the next move block is an incremental

When the following commands are incremental commands, operation is the same as when the switch is OFF.

- Manual operation during cutter or tool nose radius compensation

- When the switch is OFF
  After manual operation is performed with the switch OFF during cutter or tool nose radius compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed.
  The amount of separation equals to the amount that was performed manually.

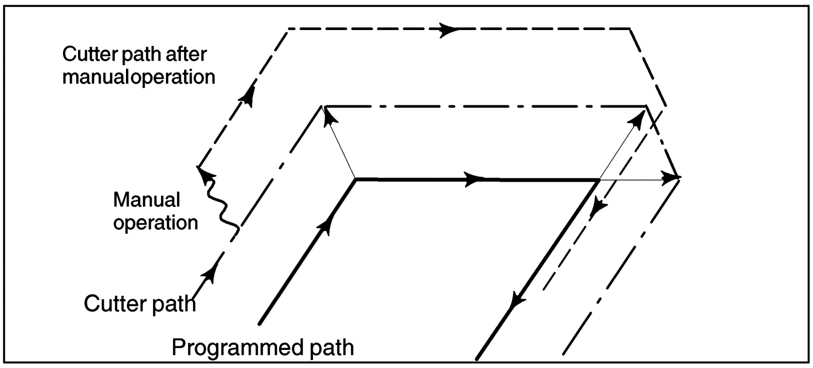


Fig. 3.5 (h)

- When the switch is ON during cutter or tool nose radius compensation
  Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the cutter or tool nose radius compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

- Manual operation performed in other than cornering
  Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B, and P_C and that the tool was manually moved to PH'. The block end point P_B moves to the point P_B' by the amount of manual movement, and vectors V_{B1} and V_{B2} at P_B also move to V_{B1}' and V_{B2}'. Vectors V_{C1} and V_{C2} between the next two blocks P_B - P_C and P_C - P_D are discarded and new vectors V_{C1}' and V_{C2}' (V_{C2}' = V_{C2} in this example) are produced from the relation between P_B' - P_C and P_C - P_D. However, since V_{B2}' is not a newly calculated vector, correct offset is not performed at block P_B' - P_C. Offset is correctly performed after P_C.

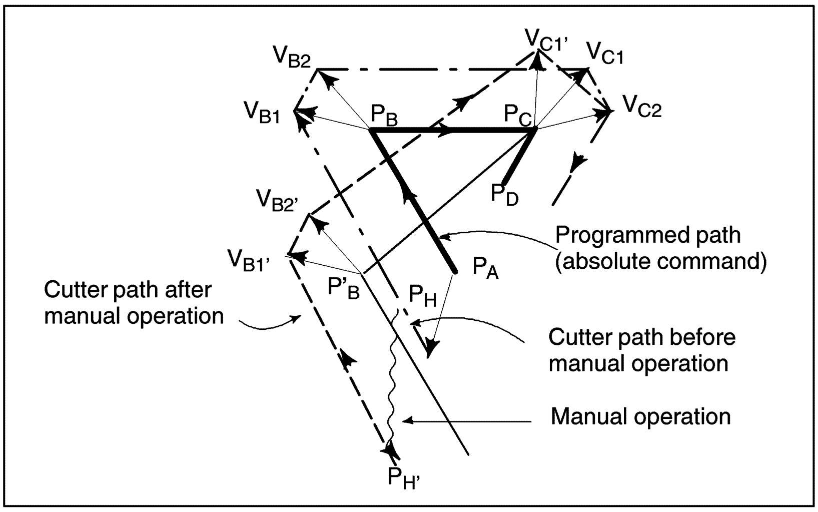


Fig. 3.5 (i)

- Manual operation during cornering
  This is an example when manual operation is performed during cornering. V_{A2}', V_{B1}', and V_{B2}' are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct cutter or tool nose radius compensation is performed for the blocks following P_C.

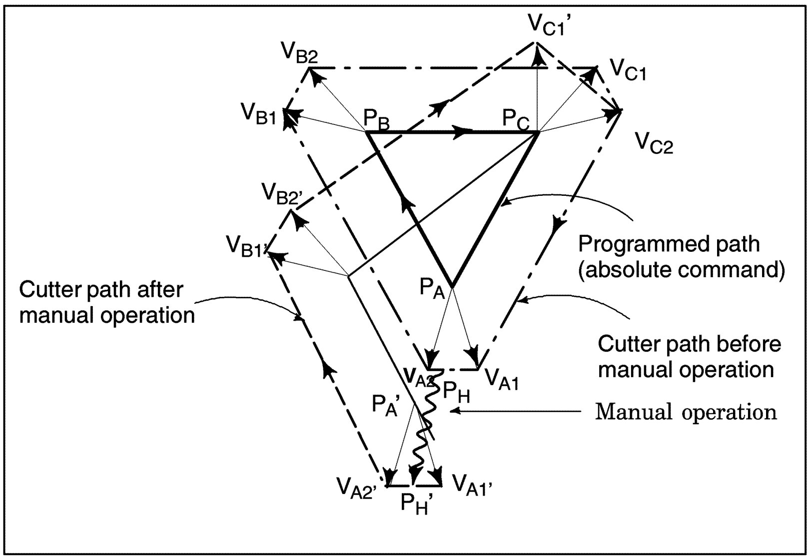
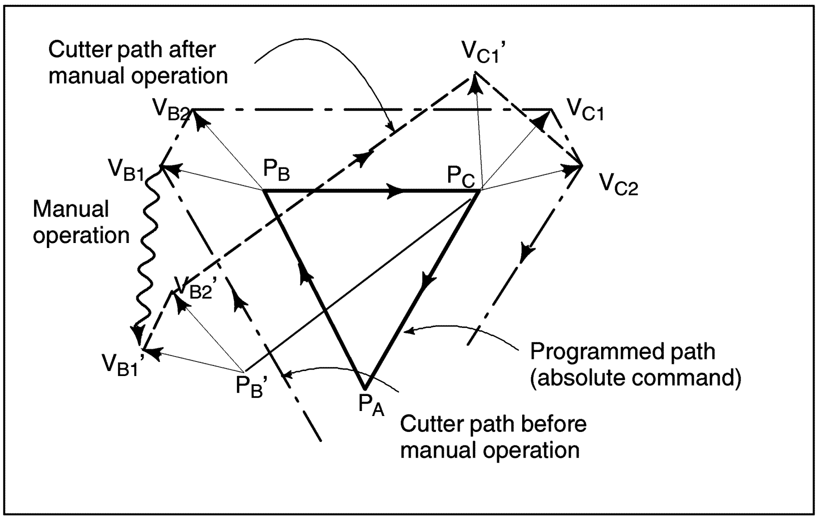


Fig. 3.5 (j)

- Manual operation after single block stop
  Manual operation was performed when execution of a block was terminated by single block stop.
  Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub-sequent processing is the same as case a described above. An MDI operation can also be intervened as well as manual operation. The movement is the same as that by manual operation.



Cutter path after
manual operation
V_{C1}'
V_{B2}
V_{C1}
V_{B1}
P_B
P_C
V_{C2}
Manual
operation
V_{B2}'
V_{B1}'
P_B'
P_A
Programmed path
(absolute command)
Cutter path before
manual operation

Fig. 3.5 (k)

3.6 RIGID TAPPING BY MANUAL HANDLE

For execution of rigid tapping, set rigid mode, then switch to handle mode and move the tapping axis with a manual handle. For rigid tapping, refer to Section 4.4 in Part II of the User's Manual (T series) or Section 5.2 in Part II of the User's Manual (M series) and the relevant manual of the machine tool builder.

Rigid tapping by manual handle

Procedure

1. Stop the spindle and servo axes, then set MDI mode by pressing the MDI switch among the mode selection switches.
2. Enter and execute the following program:
   Example 1) M series / T series (G code systems B, C)
   M29 S100 ;
   G91 G84 Z0 F1000 ;
   Example 2) T series (G code system A)
   M29 S100 ;
   G84 W0 F1000 ;
   The program above is required to determine a screw lead and set rigid tapping mode. In this program, a tapping axis must always be specified. Specify a value that does not operate the tapping axis. With the M series, specify G84 (G74). With the T series, specify G84 (G88).

> ⚠ **WARNING**
> In this MDI programming, never specify commands to position the tool at a drilling position and at point R. Otherwise, the tool moves along an axis.

3. When the entered program is executed, rigid tapping mode is set.
4. After rigid mode is set upon completion of MDI program execution, switch to the handle mode by pressing the handle switch among the mode selection switches.

> ⚠ **CAUTION**
> At this time, never press the reset key. Otherwise, rigid mode is canceled.

5. To perform rigid tapping, select a tapping axis with the handle feed axis select switch, and move the tapping axis with the manual handle.

Explanation

- Manual rigid tapping

Manual rigid tapping is enabled by parameter HRG (No. 5203#0) to 1.

- Cancellation of rigid mode

To cancel rigid mode, specify G80 as same the normal rigid tapping. When the reset key is pressed, rigid mode is canceled, but the canned cycle is not canceled.
When the rigid mode switch is to be set to off for rigid mode cancellation (when parameter CRG (No. 5200#2) is set to 0), the G80 command ends after the rigid mode switch is set to off.

- Spindle rotation direction

The rotation direction of the spindle is determined by a specified tapping cycle G code and the setting of parameter HRM (No. 5203#1). For example, when the HRM parameter is set to 0 in G84 mode, the spindle makes forward rotations as the tapping axis moves in the minus direction. (When the tapping axis moves in the plus direction, the spindle makes reverse rotations.)

- Arbitrary tapping axis

With the M series, an arbitrary tapping axis can be selected by setting bit 0 (FXY) of parameter No. 5101 to 1. With the T series, an arbitrary tapping axis can be selected not by using bit 0 (FXY) of parameter No. 5101 (but by using bit 0 (FXY) of parameter No. 5101 when the FS15 format is specified). In this case, specify a G code for plane selection and the address of the tapping axis when setting the rigid tapping mode in the MDI mode.

- G84 command/same block specifying both M29 and G84

As the MDI program command for setting the rigid mode, G84 can be specified as a rigid tapping G code (by setting bit 0 (G84) of parameter No. 5200 to 1), or M29 and G84 can be specified in the same block.
Example 1: G91 G84 Z0 F1000 S100 ;
Example 2: G91 G84 Z0 F1000 M29 S100 ;

- Specification of manual handle feed faster than the rapid traverse rate

Set parameter (No.7117) to 0 so that when manual handle feed is specified which is faster than the rapid traverse rate, the handle pulses beyond the rapid traverse rate are ignored.

- Series 15 format command

By setting bit 1 (FCV) of parameter No. 0001 to 1, the Series15 format can be used for specification.
Example 1: G91 G84.2(G84.3) Z0 F1000 S100; (M series)
Example 2: G84.2 W0 F1000 S100; (T series : G code systems B, C)
Example 3: G91 G84.2 Z0 F1000 S100; (T series : G code system A)

- Acceleration/deceleration type

When manual rigid tapping is executed, the acceleration/deceleration type and acceleration/deceleration time constant set in the rigid tapping parameters are valid.
The same settings are valid also for extraction.

- In the case of multi-spindle operation

In the case of multi-spindle operation, a spindle can be selected by specifying a P command as well as an S command.
Example : When the 2nd spindle is selected:
M29 S100 **P2**;
G91 G84 Z0 F1000 ;

Limitation

- Excessive error check

In manual rigid tapping, only an excessive error during movement is checked.

- Tool axis direction handle feed

Tool axis direction handle feed is disabled.

- Extraction override

In manual rigid tapping, the extraction override function is disabled, and the use of an acceleration/deceleration time constant for extraction is disabled.

- Number of repeats

In MDI programming, never specify K0 and L0, which are used to specify that the number of repeats is 0 and to disable the execution of a G84 block. If K0 or L0 is specified, rigid mode cannot be set.

- Positioning of the tool to a drilling position

When positioning the tool to a drilling position, select the X-axis or Y-axis with the axis select switch in handle mode. Never use the method of positioning to a drilling position in MDI mode or MEM mode. The method can operate the tapping axis.

- Three-dimensional rigid tapping

Three-dimensional rigid tapping cannot be used in rigid tapping using the manual handle.

- Interpolation type rigid tapping

Interpolation type rigid tapping cannot be used in rigid tapping using the manual handle. To perform rigid tapping using the manual handle, select the conventional type of rigid tapping with bit 3 (CHR) of parameter No. 5202.

3.7 MANUAL NUMERICAL COMMAND

The manual numerical command function allows data programmed through the MDI to be executed in jog mode. Whenever the system is ready for jog feed, a manual numerical command can be executed. The following eight functions are supported:

(1) Positioning (G00)
(2) Linear interpolation (G01)
(3) Automatic reference position return (G28)
(4) 2nd/3rd/4th reference position return (G30)
(5) M codes (miscellaneous functions)
(6) S codes (spindle speed functions)
(7) T codes (tool functions)
(8) B codes (second auxiliary functions)

By setting the following parameters, the commands for axial motion and the M, S, T, and B functions can be disabled:

(1) Positioning (G00): ....Parameter JAXx (No. 7010#0)
(2) Linear interpolation (G01):............Parameter JAXx (No. 7010#0)
(3) Automatic reference position return (G28):
......................................Parameter JAXx (No. 7010#0)
(4) 2nd/3rd/4th reference position return (G30):
......................................Parameter JAXx (No. 7010#0)
(5) M codes (miscellaneous functions): Parameter JMF (No. 7002#0)
(6) S codes (spindle speed functions):.... Parameter JSF (No. 7002#1)
(7) T codes (tool functions):................... Parameter JSF (No. 7002#2)
(8) B codes (second auxiliary functions):Parameter JBF (No. 7002#3)

Procedure Manual numerical command

Procedure

1 Press the jog switch (one of the mode selection switches).

2 Press function key PROG .

3 Press soft key [JOG] on the screen. The following manual numerical command screen is displayed.

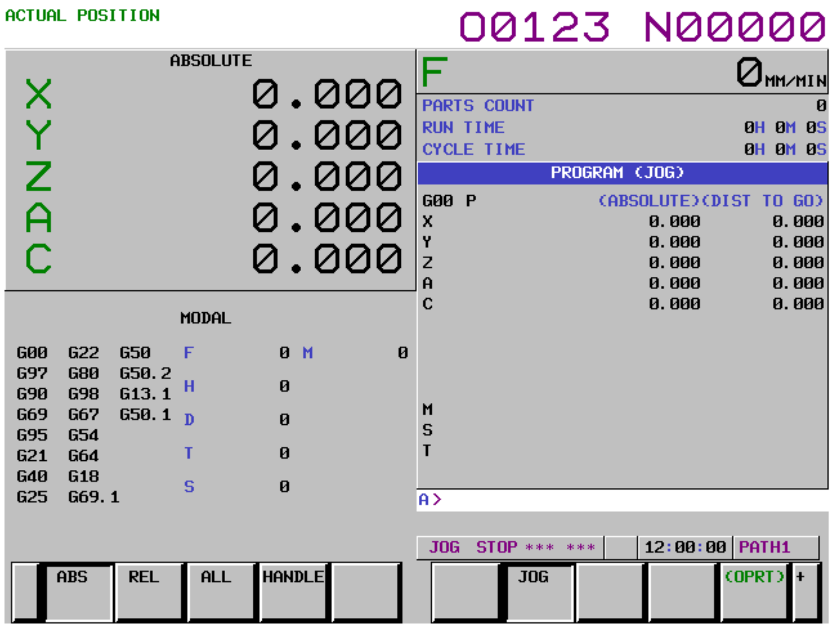


Fig. 3.7 (a) Manual numerical command screen

The remaining portion of the axis information currently not shown on the screen can be displayed by pressing the  or  key.

> **NOTE**
> 1 The actual feedrate (F) and the actual spindle speed (S) are displayed only for 9" window.
> 2 If two or more identical axis names exist for the same path, they are not displayed.

4 Enter the required commands by using address keys and numeric keys on the MDI panel, then press soft key [INPUT] or the INPUT key to set the entered data.

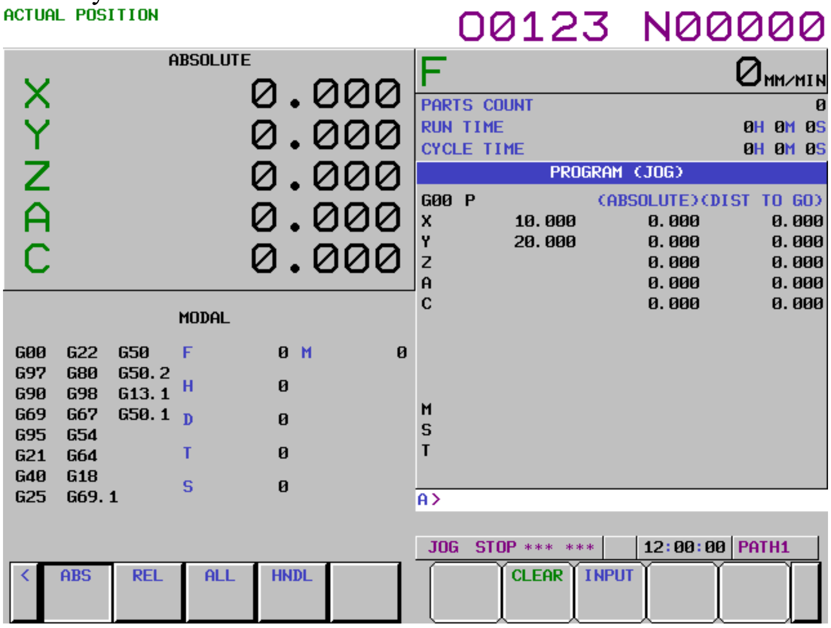


Fig. 3.7 (b)

The following data can be set:

1. G00: .... Positioning
2. G01: .... Linear interpolation
3. G28: .... Automatic reference position return
4. G30: .... 2nd/3rd/4th reference position return
5. M codes: .... Miscellaneous functions
6. S codes:........... Spindle speed functions
7. T codes:........... Tool functions
8. B codes: .... Second auxiliary functions

The set data is maintained even when the screen or mode is changed.

> **NOTE**
> When an alarm state exists, data cannot be set.

5 Press the cycle start switch on the machine operator's panel to start command execution. The status is indicated as "MSTR." The automatic operation signal, STL, can be turned on by setting parameter JST (No.7001#2).

> **NOTE**
> If the cycle start switch is pressed while an alarm state exists, a "START REJECTED (ALARM EXISTED)" warning is generated, and the entered data cannot be executed.

6 Upon the completion of execution, the "MSTR" status indication is cleared from the screen, and automatic operation signal STL is turned off. The set data is cleared entirely. G codes are set to G00 or G01 according to the setting of parameter G01 (No.3402#0).

Explanation

- Positioning

An amount of travel is given as a numeric value, preceded by an address such as X, Y, or Z. This is always regarded as being an incremental command, regardless of whether G90 or G91 is specified.

	Manual rapid traverse selection switch	
	Off	**On**
Feedrate (parameter)	Jog feedrate for each axis (No.1423)	Rapid traverse rate for each axis (No.1420)
Automatic acceleration/ deceleration (parameter)	Exponential acceleration/ deceleration in jog feed for each axis (No.1624)	Linear acceleration/deceleration in rapid traverse for each axis (No.1620)
Override	Manual feed override	Rapid traverse override

NOTE

1 When the manual rapid traverse switch is off, the feedrate is clamped so that the axis-by-axis manual rapid traverse rate (parameter No. 1424) is not exceeded.

2 The tool cannot move if linear interpolation type positioning (where the tool path is linear) has been performed by setting parameter LRP (No. 1401 #1).

- Linear interpolation (G01)

An amount of travel is given as a numeric value, preceded by an address such as X, Y, or Z. This is always regarded as being an incremental command, regardless of whether G90 or G91 is specified. Axial movements are always performed in incremental mode even during scaling or polar coordinate interpolation. In addition, movement is always performed in feed per minute mode regardless of the specification of G94 or G95.

Feedrate (parameter)	Dry run feedrate (No. 1410)
Automatic acceleration/deceleration (parameter)	Exponential acceleration/deceleration in cutting feed for each axis (No.1622)
Override	Manual feed override

> **NOTE**
> Since the feedrate is always set to the dry run feedrate, regardless of the setting of the dry run switch, the feedrate cannot be specified using F. The feedrate is clamped such that the maximum cutting feedrate, set in parameter No. 1430, is not exceeded.

- Automatic reference position return (G28)

The tool returns directly to the reference position without passing through any intermediate points, regardless of the specified amount of travel. For axes for which no move command is specified, however, a return operation is not performed.

Feedrate (parameter)	Rapid traverse rate (No. 1420)
Automatic acceleration/deceleration (parameter)	Linear acceleration/deceleration in rapid traverse for each axis (No. 1620)
Override	Rapid traverse override

- 2nd, 3rd, or 4th reference position return (G30)

The tool returns directly to the 2nd, 3rd, or 4th reference position without passing through any intermediate points, regardless of the specified amount of travel. To select a reference position, specify 2, 3, or 4 in address P. If address P is omitted, a return to the second reference position is performed.

Feedrate (parameter)	Rapid traverse rate (No. 1420)
Automatic acceleration/deceleration (parameter)	Linear acceleration/deceleration in rapid traverse for each axis (No. 1620)
Override	Rapid traverse override

> **NOTE**
> The function for 3rd/4th reference position return is optional.
> 1 When the option is not selected
> If "P2" is not specified in address P, the alarm PS0046 is generated and the function cannot be executed.
> 2 When the option is selected
> If "P2," "P3," or "P4" is not specified in address P, the alarm PS0046 is generated and the function cannot be executed.

- M codes (Auxiliary functions)

After address M, specify a numeric value of no more than the number of digits specified by parameter No. 3030. When M98 or M99 is specified, it is executed but not output to the PMC.

> **NOTE**
> Neither subprogram calls nor custom macro calls can be performed using M codes.

- S codes (spindle speed functions)

After address S, specify a numeric value of no more than the number of digits specified by parameter No. 3031.

> **NOTE**
> Subprogram calls cannot be performed using S codes.

- T codes (tool functions)

After address T, specify a numeric value of no more than the number of digits specified by parameter No. 3032.

> **NOTE**
> Subprogram calls cannot be performed using T codes.

- B codes (second auxiliary functions)

After address B, specify a numeric value of no more than the number of digits specified by parameter No. 3033.

> **NOTE**
> 1 B codes can be renamed "U," "V," "W," "A," or "C" by setting parameter No. 3460. If the new name is the same as an axis name address, "B" is used. Note that "U," "V," and "W" can be used for the T codes only when the G codes are B or C.
> 2 Subprogram calls cannot be performed using B codes.

- Data input

(1) When addresses and numeric values of a command are typed, then soft key [INPUT] is pressed, the entered data is set. In this case, the input unit is either the least input increment or calculator-type input format, according to the setting of bit 0 (DPI) of parameter No. 3401.

The INPUT key on the MDI panel can be used instead of soft key [INPUT].

(2) Commands can be typed successively.

(3) Key entry is disabled during execution.
If soft key [INPUT] or the INPUT key on the MDI panel is pressed during execution, an "EXECUTION/MODE SWITCHING IN PROGRESS" warning is output.

(4) If input data contains an error, the following warnings may appear:

Warning	Description
FORMAT ERROR	- A G code other than G00, G01, and G28 has been entered. - An address other than those displayed on the manual numerical command screen has been entered. A value that exceeds the following limitations has been entered. - Address G: .... 2 digits - Address P:.......... 1 digit - Axis address:...... 9 digits - M, S, T, B: .... The parameter-set number of digits

NOTE
Even when the memory protection key is set, key input can nevertheless be performed.

- Erasing data

(1) When soft key [CLEAR] is pressed, followed by soft key [EXEC], all the set data is cleared. In this case, however, the G codes are set to G00 or G01, depending on the setting of bit 0 (G01) of parameter No. 3402.
Data can also be cleared by pressing the RESET key on the MDI panel.

(2) If soft key [CLEAR] is pressed during execution, an "EXECUTION/MODE SWITCHING IN PROGRESS" warning is output.

- Scrolling pages

If the whole axis information cannot be displayed on a single page, you can scroll pages by pressing the PAGE ↑ or PAGE ↓ key.

- Halting execution

If one of the following occurs during execution, execution is halted, and the data is cleared in the same way as when soft key [CLEAR] is pressed. The remaining distance to be traveled is canceled.
(1) When a feed hold is applied
(2) When the mode is changed to other than jog feed mode
(3) When an alarm is generated
(4) When a reset or emergency stop is applied
The M, S, T, and B functions remain effective even upon the occurrence of the above events, with the exception of (4).

- Modal information

Modal G codes and addresses used in automatic operation or MDI operation are not affected by the execution of commands specified using the manual numerical command function.

- Jog feed

When the tool is moved along an axis using a feed axis and direction selection switch on the manual numerical command screen, the remaining amount of travel is always shown as "0".

- Disabling the M, S, T, and B functions

By setting bit 0 to bit 3 (JMF, JSF, JTF, and JBF) of parameter No. 7002, the M, S, T, and B functions can be disabled. If a disabled function is specified, the warning "THIS COMMAND CANNOT BE EXECUTED." is issued.

Limitation

- M, S, T, and B functions

While automatic operation is halted, manual numerical commands can be executed. In the following cases, however, a "START REJECTED (ALREADY EXECUTING)" warning is output, and command execution is disabled.

(1) When an M, S, T, or B function is already being executed, a manual numerical command containing an M, S, T, or B function cannot be executed.

(2) When an M, S, T, or B function is already being executed, and that function alone is specified or a block specifying that function also contains another function (such as a move command or dwell function) which has already been completed, a manual numerical command cannot be executed.

- Jog feed

When a manual numerical command is specified while the tool is being moved along an axis by using a feed axis and direction selection switch, the axial movement is interrupted, and the manual numerical command is executed. Therefore, the tool cannot be moved along an axis by using a feed axis and direction selection switch during execution of a manual numerical command.

- Mirror image

A mirror image cannot be produced for the direction of a specified axial movement.

- REF mode

The manual numerical command screen is not displayed in the REF mode.

- Indexing of the index table and chopping

Commands cannot be specified for an axis along which operation is being performed during indexing or chopping.
If such an axis is specified for execution, a "THIS COMMAND CAN NOT EXECUTE" warning is output.

- Functions that cannot be specified

Commands cannot be specified for axes that operate using any of the following functions.

- Index table indexing
- Chopping
- Spindle positioning
- Polygon turning
- Axis recomposition and superposition control

If the commands are executed for any such axis, a "THIS COMMAND CAN NOT EXECUTE" warning is generated.

- Functions that cannot be used

Commands cannot be specified for the following functions.

- Extended axis name
- Extended spindle name
- Address P command for multi spindle
- Cs contour control function

3.8 MANUAL FEED FOR 5-AXIS MACHINING

This function enables the use of the following functions.

- Manual feed for 5-axis machining
  - Tool axis direction handle feed/tool axis direction JOG feed/tool axis direction incremental feed
  - Tool axis right-angle direction handle feed/tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed
  - Tool tip center rotation handle feed/tool tip center rotation JOG feed/tool tip center rotation incremental feed
  - Table vertical direction handle feed/table vertical direction JOG feed/table vertical direction incremental feed
  - Table horizontal direction handle feed/table horizontal direction JOG feed/table horizontal direction incremental feed

  A handle interrupt can be generated for each handle feed. Handle interrupts work according to the corresponding handle feed specifications described hereinafter unless otherwise noted.

- Screen display functions
  - Display of the coordinate of the tool tip
  - Display of pulse values
  - Display of the amount of machine axes movement

NOTE

1 To execute 5-axis machining handle feed requires the manual handle feed option. Also, to generate 5-axis machining handle interrupts requires the manual handle interrupt option.

2 A 5-axis machining handle interrupt must not be generated when a rotation axis command is being executed during automatic operation.

3 Manual feed for 5-axis machining is disabled when the manual reference position return mode is selected.

3.8.1 Tool Axis Direction Handle Feed/Tool Axis Direction JOG Feed/Tool Axis Direction Incremental Feed

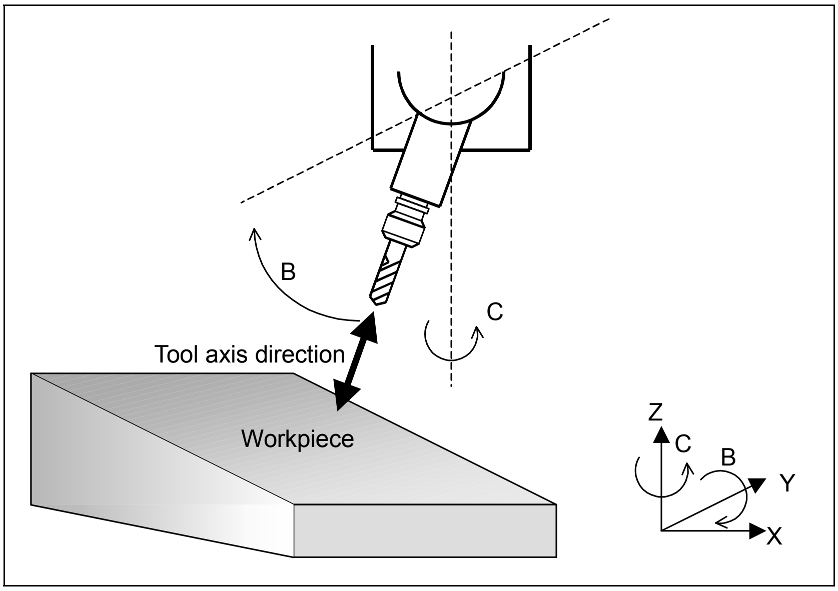
Overview

In the tool axis direction handle feed, tool axis direction JOG feed, and tool axis direction incremental feed, the tool or table is moved in the tool axis direction.

Explanation

- Tool axis direction

The tool axis direction that is taken when all the rotation axes for controlling the tool are at an angle of 0 degree is specified in parameters No.19697, No.19698, and No.19699. As the rotation axes for controlling the tool rotate, the tool axis direction changes according to the rotation axis angle.



- Tool axis direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis direction feed in the tilted working plane command mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane command.

- Tool axis direction handle feed

The tool axis direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis direction handle feed mode effective is set in parameter No.12310.
<4> The value of parameter No.12310 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool axis direction JOG feed/tool axis direction incremental feed

The tool axis direction JOG feed or tool axis direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction specified by parameter No.19697. (Even when the tool axis direction is slant because of the settings of parameters No.19698 and No.19699, the signal that activates the tool axis direction JOG feed or tool axis direction incremental feed is determined by parameter No.19697 only.)

Ex.) No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.

- +J3: Tool axis direction +
- -J3: Tool axis direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

3.8.2 Tool Axis Right-Angle Direction Handle Feed/Tool Axis Right-Angle Direction JOG Feed/Tool Axis Right-Angle Direction Incremental Feed

Overview

In the tool axis right-angle direction handle feed, tool axis direction JOG feed, or tool axis direction incremental feed, the tool or table is moved in the tool axis direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the tool axis direction vector.

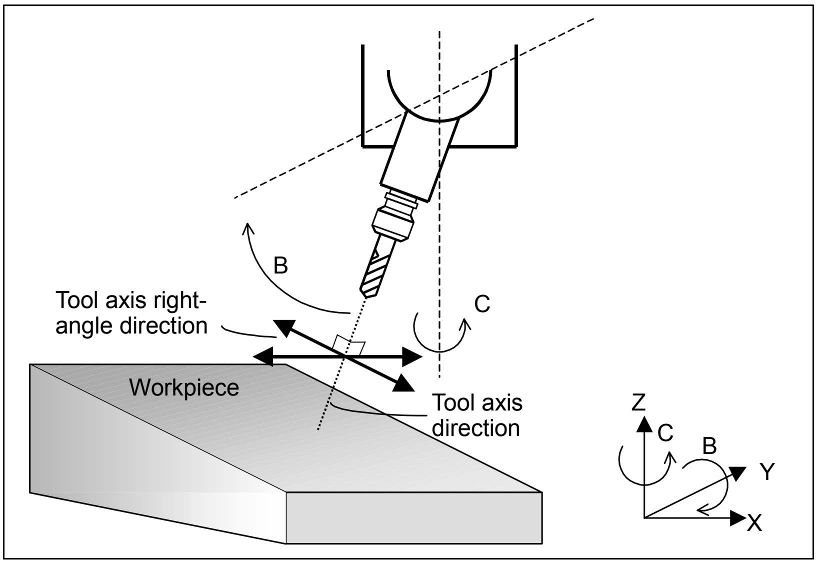
Explanation

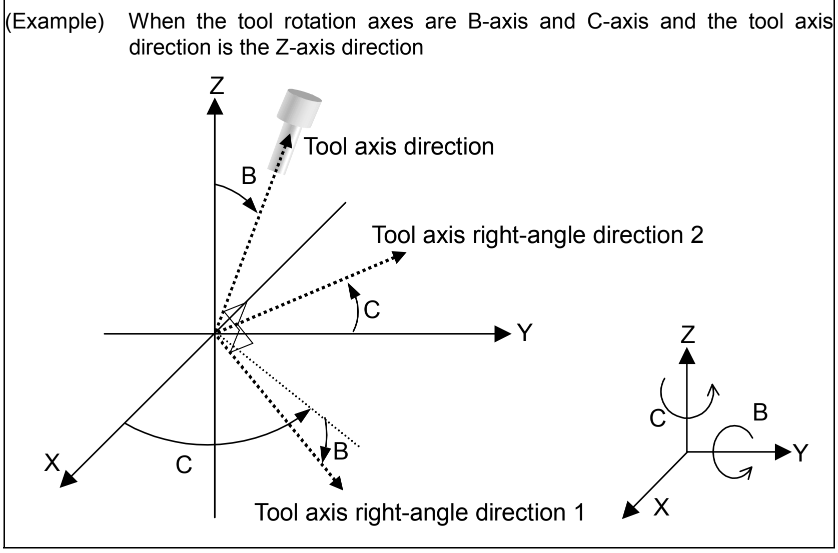
- Tool axis right-angle direction

There are two tool axis right-angle directions, which are perpendicular to the tool axis direction (see the previous section).

Parameter No.19697	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the tool axis right-angle directions that may be taken when the angles of all the rotation axes for controlling the tool are 0 degree and when parameters No.19698 and No.19699 are both set to 0.
When the reference tool axis direction is inclined based on the settings of parameters No.19698 and No.19699, the tool axis right-angle direction is also inclined as much.
As the rotation axes for controlling the tool rotate, the tool axis right-angle direction changes according to the rotation axis angle.





- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the tool axis direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the tool axis right-angle direction 1 (longitude direction) vector ($\vec{R1}$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

Equation: $\vec{R1} = \vec{P} \times \vec{T}$

Let a vector perpendicular to the tool axis direction vector ($\vec{T}$) and tool axis right-angle direction 1 (longitude direction) vector ($\vec{R1}$) be the tool axis right-angle direction 2 (latitude direction) vector ($\vec{R2}$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a

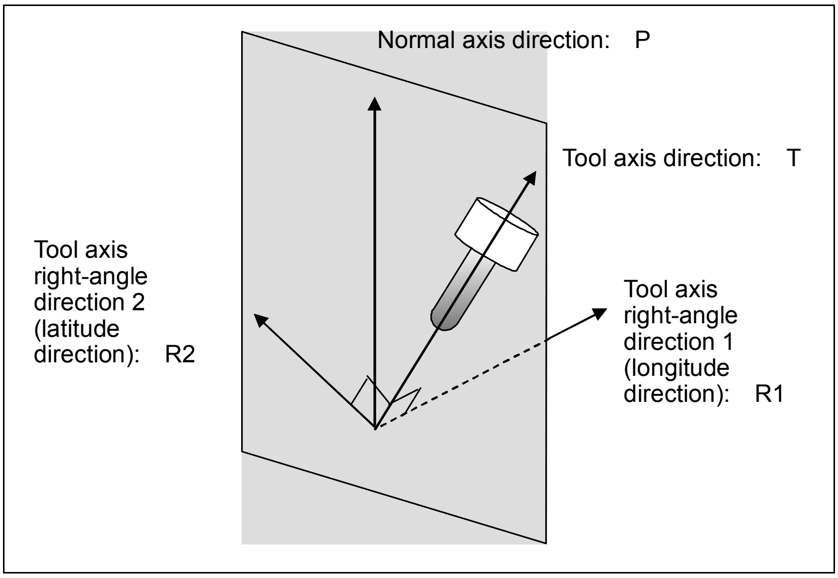
movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

Equation: $\vec{R}2 = \vec{T} \times \vec{R}1$

When the tool axis direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Parameter No.12321	Normal axis direction	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the reference tool axis direction (parameter No. 19697).
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459 is issued.



- Tool axis right-angle direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis right-angle direction feed in the tilted working plane command mode is defined as follows:

Tool axis right-angle direction 1: X direction in the feature coordinate system of the tilted working plane command

Tool axis right-angle direction 2: Y direction in the feature coordinate system of the tilted working plane command

- Tool axis right-angle direction handle feed

The tool axis right-angle direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis right-angle direction handle feed mode effective is set in parameter No.12311 or No.12312.
<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A - HS1E).

- Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis right-angle direction by the amount of rotation.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed

The tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697. (Even when the tool axis direction is slant because of the settings of parameters No.19698 and No.19699, the signal that activates the tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is determined by parameter No.19697 only.)

Ex.) No.19697=3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1: Tool axis right-angle direction 1 +
- -J1: Tool axis right-angle direction 1 -
- +J2: Tool axis right-angle direction 2 +
- -J2: Tool axis right-angle direction 2 -

- Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

- Feedrate clamp

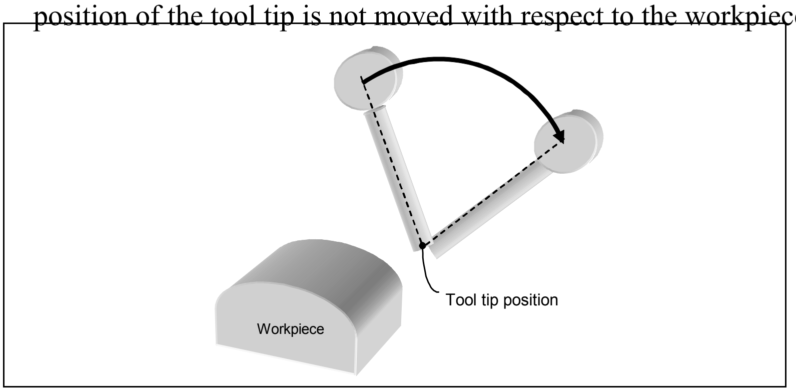
The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

3.8.3 Tool Tip Center Rotation Handle Feed/Tool Tip Center Rotation JOG Feed/Tool Tip Center Rotation Incremental Feed

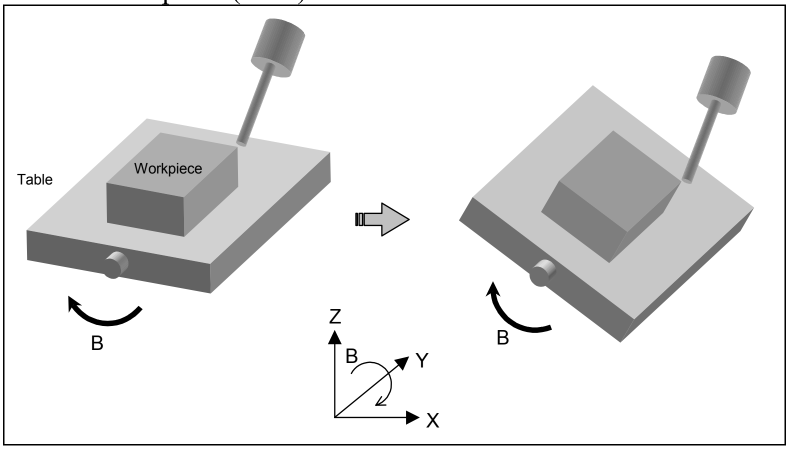
Overview

In the tool tip center rotation handle feed, tool tip center rotation JOG feed, and tool tip center rotation incremental feed, when a rotary axis is rotated by manual feed, the linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece (table).

- The following figure shows an example where the tool is rotated on the rotation axis. In this case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece.



- The following figure shows an example where the table is rotated on the rotation axis. As in the previous case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece (table).



- Tool tip center rotation handle feed

The tool tip center rotation handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.

<2> The tool tip center rotation feed mode signal (RNDH) is set to "1".

<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool tip center rotation handle feed mode effective is set in parameter No.12313 or No.12314.

<4> The value of parameter No.12313 or No.12314 matches the first manual handle feed axis selection signals (HS1A - HS1E).

- Amount of movement

When the manual pulse generator is rotated, the rotation axis is moved by the amount of rotation. The linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece.

- Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool tip center rotation JOG feed/tool tip center rotation incremental feed

The tool tip center rotation JOG feed or tool tip center rotation incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> The tool tip center rotation feed mode signal (RNDH) is set to "1".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the rotation axis to be rotated.

Ex.) When the B-axis (4th axis) is rotated

- +J4: Tool tip center rotation feed +
- -J4: Tool tip center rotation feed -

- Feedrate

Control is exerted so that the synthetic speed of the linear axes (in the tangential direction) is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

- Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis).

- Selection of the tool length offset value

The tool length in manual feed for 5-axis machining is determined as explained below. Table 3.8.3 (a))
If bit 2 (LOD) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is performed, the offset data specified for the tool length offset is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is not performed, the tool length is determined as follows. If bit 3 (LOZ) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length in manual feed for 5-axis machining; if LOZ is set to 1, the tool length is assumed to be 0.

Table 3.8.3 (a) Tool length offset value in manual feed for 5-axis machining

<table>
<tr><th colspan="2" rowspan="3"></th><th colspan="3">19746#2(LOD)</th></tr>
<tr><th rowspan="2">= 0</th><th colspan="2">= 1</th></tr>
<tr><th>Tool length offset enabled</th><th>Tool length offset canceled</th></tr>
<tr><td rowspan="2">19746#3 (LOZ)</td><td>= 0</td><td rowspan="2">Parameter No. 12318</td><td rowspan="2">Offset data</td><td>Parameter No. 12318</td></tr>
<tr><td>= 1</td><td>0</td></tr>
</table>

The tool length offset function is enabled when the following two conditions are both satisfied:

- The tool length offset function listed below is enabled (modal code of group 8 except G49)
  - G43／G44 : Tool length compensation
  - G43.4／G43.5 : Tool center point control
- The H/D code is other than 0.

If bit 6 (CLR) of parameter No. 3402 is set to 0 not to clear the tool length offset vector, G codes of group 8, and H codes at the time of a reset, the tool length offset status is maintained when a reset is made in the tool length offset mode.

3.8.4 Table Vertical Direction Handle Feed/Table Vertical Direction JOG Feed/Table Vertical Direction Incremental Feed

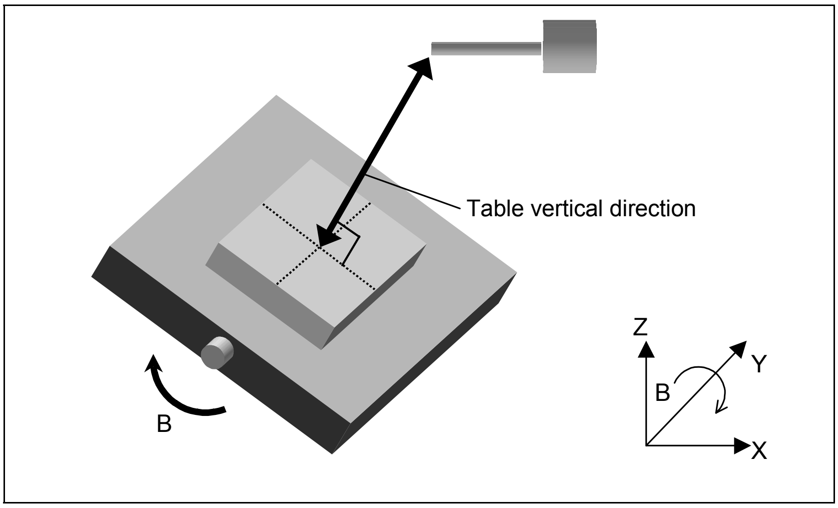
Overview

In the table vertical direction handle feed, table vertical direction JOG feed, and table vertical direction incremental feed, the tool is moved in the table vertical direction.

Explanation

- Table vertical direction

The table vertical direction is a direction vertical to the table. It is equal to the tool axis direction specified in parameter No.19697 when all of the rotation axes for controlling the table are at a an angle of 0 degree. When the rotation axes for controlling the table rotate, the table vertical direction changes according to the rotation axis angle.



- Table-based vertical direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based vertical direction feed in the tilted working plane command mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane command.

- Table vertical direction handle feed

The table vertical direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.

<2> Both the tool axis direction feed mode signal (ALNGH) and the table base signal (TB_BASE) are set to "1".

<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the table vertical handle feed mode effective is set in parameter No.12310.

<4> The value of parameter No.12310 matches the first manual handle feed axis selection signals (HS1A - HS1E).

- Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table vertical direction by the amount of rotation.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Table vertical direction JOG feed/table vertical direction incremental feed

The table vertical direction JOG feed or table vertical direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> Both the tool axis direction feed mode signal (ALNGH) and the table base signal (TB_BASE) are set to "1".

<3> The feed axis direction selection signal [+Jn,-Jn (where n = 1 to the number of controlled axes)] is set to "1" for the axis corresponding to the direction specified by parameter No.19697.

Ex.) No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.

- +J3: Table vertical direction +
- -J3: Table vertical direction -

- Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

3.8.5 Table Horizontal Direction Handle Feed/Table Horizontal Direction JOG Feed/Table Horizontal Direction Incremental Feed

Overview

In the table horizontal direction handle feed, table horizontal direction JOG feed, and table horizontal direction incremental feed, the tool is moved in the table horizontal direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the table-based vertical direction vector.

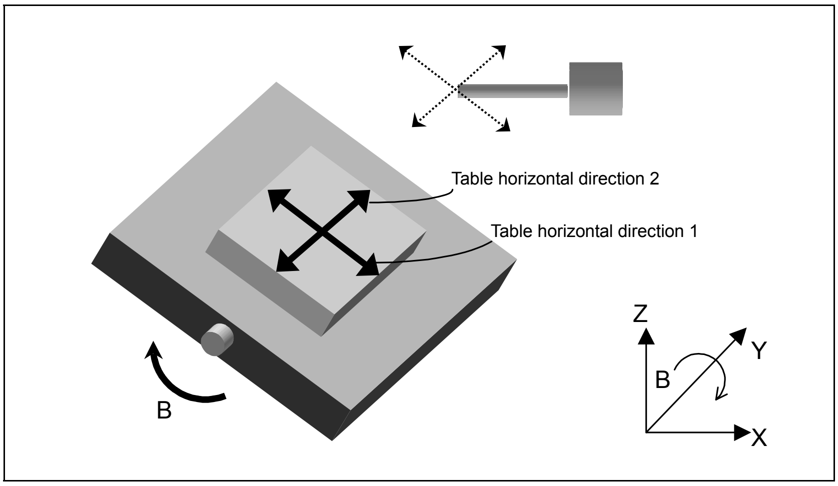
Explanation

- Table horizontal direction

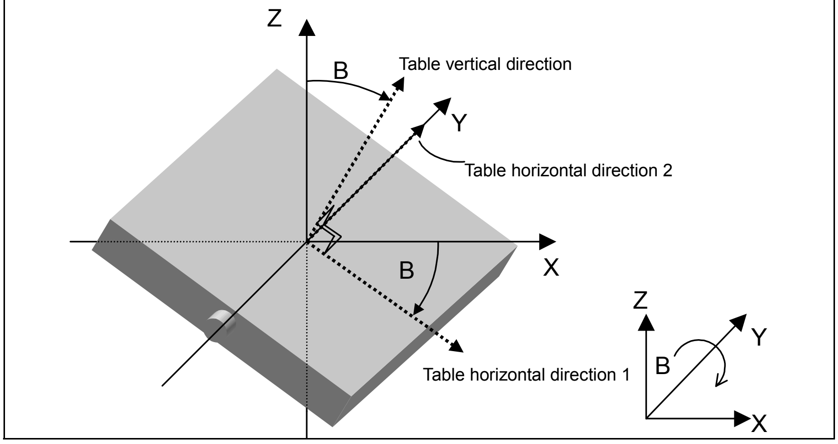
There are two table horizontal directions, which are perpendicular to the table vertical direction (see the previous section).

Parameter No.19697	Table horizontal direction 1	Table horizontal direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the table horizontal directions that may be taken when the angles of all the rotation axes for controlling the table are 0 degree.
As the rotation axes for controlling the table rotate, the table horizontal direction changes according to the rotation axis angle.



(Example) When the table rotation axis is the B-axis, and the table vertical direction is the Z-axis direction



- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the table-based vertical direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the table-based horizontal direction 1 (longitude direction) vector ($\vec{R}1$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

Equation: $\vec{R}1 = \vec{P} \times \vec{T}$

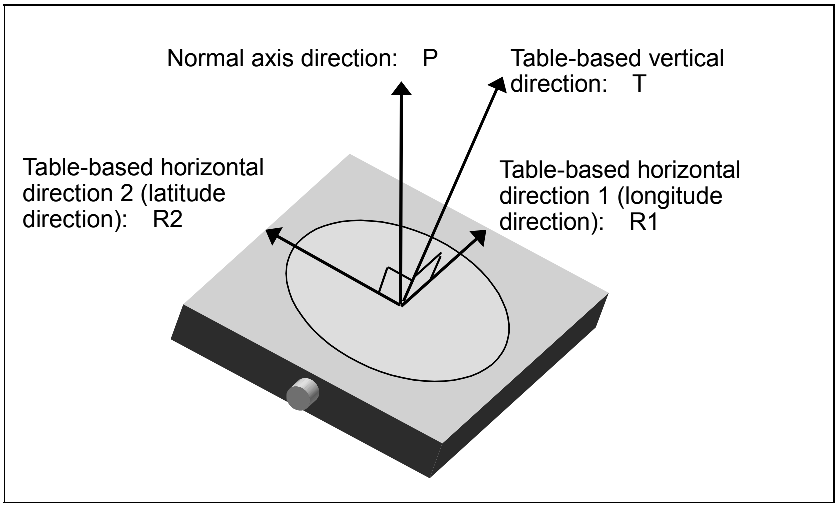
Let a vector perpendicular to the table-based vertical direction vector ($\vec{T}$) and table-based horizontal direction 1 (longitude direction) vector ($\vec{R}1$) be the table-based horizontal direction 2 (latitude direction) vector ($\vec{R}2$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

Equation: $\vec{R}2 = \vec{T} \times \vec{R}1$

When table-based vertical direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Parameter No. 12321	Normal axis direction	Table-based horizontal direction 1	Table-based horizontal direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the tool axis direction.
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459 is issued.



- Table-based horizontal direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based horizontal direction feed in the tilted working plane command mode is defined as follows:

Table-based horizontal direction 1: X direction in the feature coordinate system of the tilted working plane command
Table-based horizontal direction 2: Y direction in the feature coordinate system of the tilted working plane command

- Table horizontal direction handle feed

The table horizontal direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> Both the tool axis right-angle direction feed mode signal (RGHTH) and the table base signal (TB_BASE) are set to 1.
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the table horizontal direction handle feed mode effective is set in parameter No.12311 or No.12312.
<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A - HS1E).

- Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table horizontal direction by the amount of rotation.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Table horizontal direction JOG feed/table horizontal direction incremental feed

The table horizontal direction JOG feed or table horizontal direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> Both the tool axis right-angle direction feed mode signal (RGHTH) and the table base signal (TB_BASE) are set to "1".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697.

Ex.) No.19697 = 3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1: Table horizontal direction 1 +
- -J1: Table horizontal direction 1 -
- +J2: Table horizontal direction 2 +
- -J2: Table horizontal direction 2 -

- Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) of a driven feed axis direction selection signal. The manual feedrate override feature is available.

- Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

Note

1 To perform a handle feed for 5-axis machining, the manual handle feed option is required. To perform a handle interrupt for 5-axis machining, the manual handle interrupt option is required.

2 When a handle interrupt for 5-axis machining is performed, rotation axis command execution must not be in progress in automatic operation.

3 When the manual reference position return mode is selected, manual feed for 5-axis machining is not enabled.

4 When the offset value specified for the tool length offset function is used for tool center point rotation feed (when bit 2 (LOD) of parameter No. 19746 is set to 1), the controlled point should generally be shifted. (Set bit 5 (SVC) of parameter No. 19665 to 1.)
In this case, specify the tool length with a radius value.

3.9 DISTANCE CODED LINEAR SCALE INTERFACE

Overview

The interval of each reference marks of distance coded linear scale are variable. Accordingly, if the interval is determined, the absolute position can be determined. The CNC measures the interval of reference marks by axis moving of short distance and determines the absolute position. Consequently the reference position can be established without moving to reference position.

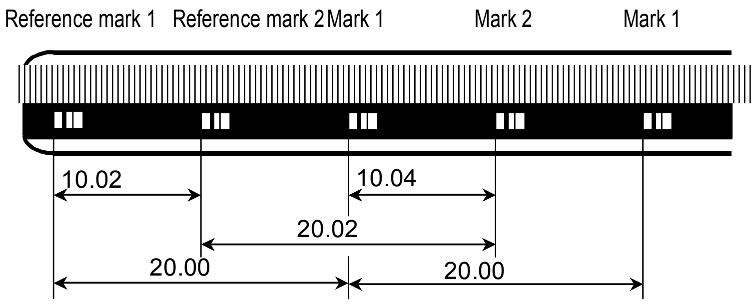


Fig.3.9(a) Example of distance coded linear scale

This is an optional function.

3.9.1 Procedure for Reference Position Establishment

Procedure

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set a direction selection signal(+J1,-J1,+J2,-J2,…) for a target axis.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter (No.1425) setting).
(4) When a reference mark is detected, the axis stops, then the axis is fed at a constant low speed again.
(5) Above (4) is executed repeatedly until two, three or four reference marks are detected. And absolute position is determined and reference position establishment signal (ZRF1,ZRF2,ZRF3, …) turns to "1".
(A number of reference marks is determined by the parameter No.1802#2, #1.)

Even when the direction selection signal (+J1, -J1, +J2, -J2,...) is set to "0" while steps (2) to (5) are being performed, the feedrate operation does not stop, and the operation for establishing a reference position is carried out continuously.

The timing chart for this procedures is given below.

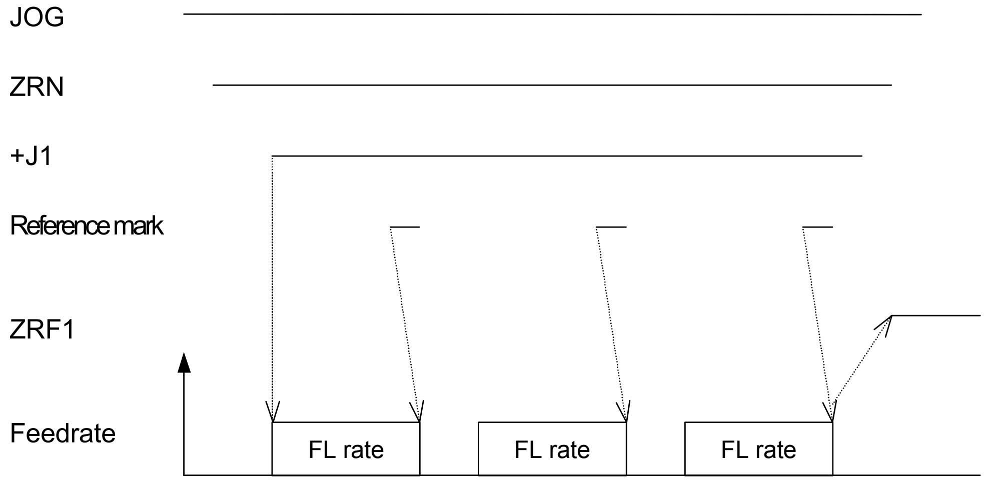


Fig.3.9.1(a) Timing chart for reference position establishment

- Procedure for establishing a reference position through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (5) above are performed automatically.
After the reference position is established, the automatic reference position return is performed.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of the following operations is performed in steps (3) to (5), described above.

- Reset
- Setting the feed axis direction selection signal (+J1, -J1, +J2, -J2, etc.) to 0

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position

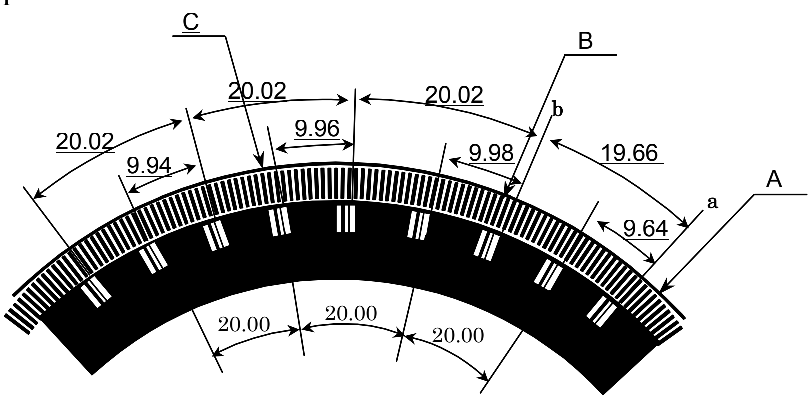
If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

3.9.2 Reference Position Return

(1) When the reference position is not established and the axis moved by turning the feed axis direction signal (+J1,-J1,+J2,-J2,...) to "1" in REF mode, the reference position establishment procedure is executed.
(2) When the reference position is already established and the axis is moved by turning the feed axis direction signal (+J1,-J1,+J2,-J2,...) to "1" in REF mode, the axis is moved to the reference point without executing the reference position establishment procedure.
(3) When the reference position is not established and the reference position return command (G28) is executed, the reference position establishment procedure is executed. The next movement the axis depends on the setting of parameter RFS (No.1818#0).
(4) When the reference position is already established and the reference position command (G28) is executed, the movement of the axis depends on the setting of parameter RF2 (No.1818#1).

3.9.3 Distance Coded Rotary Encoder

In case of setting a rotary axis, if a parameter DCRx (No. 1815#3) is set, the setting axis is regarded as being equiped with a distance coded rotary encoder.
In case of distance coded rotary encoder, the marker interval may be different from parameter setting value. (a-b section of the following figure) When the reference point return is executed through this section, it is not able to establish the reference point. Therefore, in case of distance coded rotary encoder, if the reference point return is started for B point from A point of below figure, the reference point is not established yet at B point. The reference point return is re-started for C point. The reference point return procedure is finished at C point.



- When the reference point return procedure is executed, the coordinate value are rounded in 0 to 360 degree, even if Machine coordinate values are linear axis type.

- In case of distance coded rotary encoder, only the measurement by three points or four points is possible. (parameter 1802#2(DC2) is disregarded as 0.)

3.9.4 Axis Synchronization Control

Requirements when this function is used with axis synchronization control axes

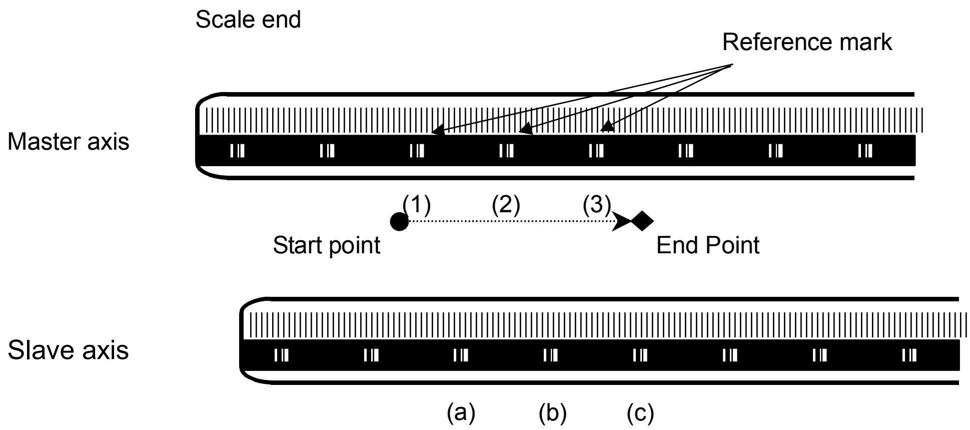
When this function is used with axis synchronization control axes, the distance coded linear scale used for the master axis and that used for the slave axis must have reference marks placed at identical intervals. (Set identical values in parameter Nos. 1821 and 1882 for both the master and slave axes.)
This function does not work unless the use of this function is specified for both the master and slave axes (bit 7 (DCL) of parameter No. 1815 is 1).
Also, in all parameters related to this function, except parameter No. 1883, 1884 (distance from the scale zero point to reference position 1, 2), set identical values for both the master and slave axes.
If a parameter value for the master axis differs from the corresponding parameter value for the slave axis, alarm SV1051 is issued.

NOTE
When this function is used with axis synchronization control axes for which the operation mode is switched between synchronization operation and normal operation, this function is enabled only if the synchronization select signal (SYNC1, SYNC2, ...) is 1. (During establishment of a reference position, the synchronization select signal status must be maintained.)

Reference position establishment with axis synchronization control axes

With axis synchronization control axes, a reference position is established as follows. When a reference mark for the master or slave axis is detected, a stop takes place temporarily. Then, a feed operation is performed again at the reference position return FL feedrate. This sequence is repeated until a reference mark is detected three or four times for both the master and slave axes. Then the absolute position is calculated for both the master and slave axes, and the reference position establish signals (ZRF1, ZRF2, ...) are set to 1.
After the reference position has been established by the above operation, a synchronization error is corrected. (Checking for excessive synchronization error alarm 2 is made even during reference position establishment.)

(Example of 3 points measurement system)



In the above example, the following sequence is executed.

a. When the reference mark (1) of the master axis is detected, both master axis and slave axis stop.
b. Both the axes begin to move again at a reference position return FL feedrate.
c. When the reference mark (a) of the slave axis is detected, both axes stop again.
d. Both the axes begin to move again at FL feedrate.
e. Both axes repeat the operation until all point ((2) -> (b) ->(3) -> (c)) are detected.
f. When the slave axis detects the third reference mark (c), both the axes end the reference position establishment.

> **NOTE**
> In case of this function is used with axis synchronization control axes, if the value of parameter No. 1883 and 1884 for both the master and slave axes is 0, the reference position is not established. Also, the reference position establish signals (ZRF1, ZRF2, ...) are set to 0.

3.9.5 Axis Control by PMC

In PMC axis control, if the reference position return command (axis control command code 05H) is issued for an axis having a distance coded linear scale, reference position return is performed according to the reference position return sequence for the distance coded linear scale. Specifically, the following operations take place:

Before reference position establishment	The reference position is established by detecting two, three or four reference marks. Movement to the reference position is not performed.
After reference position establishment	Positioning at the reference position is performed.

3.9.6 Angular Axis Control

There are the following limitations when the angular axis control is used.

(a) It is necessary to use the linear scale with the distance coded reference mark for both the perpendicular axis and the angular axis.

(b) When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the alarm DS0020 is generated.

(c) During the reference point establishment operation of the angular axis, the command in the perpendicular axis is invalid in the manual reference point return.

3.9.7 Note

(1) In the case of the actual interval of reference marks is different from parameter setting value, the alarm DS1449 occurs.

(2) This function is disabled if any of the following conditions is satisfied:
- Either parameter 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0.
- Parameters 1821 and 1882 have identical settings.
- The difference between the settings made for parameters 1821 and 1882 is greater than or equal to twice either setting.
- The absolute-position detection function is enabled.

(3) A difference of parameter No.1821 and No.1882 must be more than 4.

Example)

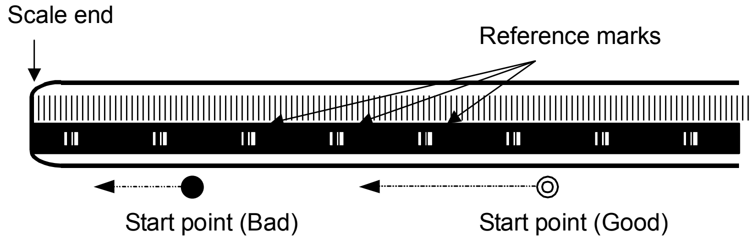
When the scale, which is that mark1 interval is 20.000mm and mark2 interval is 20.004mm, is used on IS-B machine :
When the detection unit of 0.001mm is selected, parameter No.1821 and No.1882 must be set "20000" and "20004", and the difference of them is "4".
To use such a scale, please adjust the detection unit by modification of parameter No.1820(CMR) and No.2084/2085(flexible feed gear) to make the difference of No.1821 and 1882 more than 4 as following examples.

(a) Set the detection unit=0.0001mm, and set No.1821=200000, No.1882=200040

(b) Set the detection unit=0.0005mm, and set No.1821=40000, No.1882=40008

> **NOTE**
> When the detection unit is changed, parameters relating to the detection unit (such as the effective area and positional deviation limit) must also be changed accordingly.

(4) In this procedure, the axis does not stop until two, three or four reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three or four reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.



(5) When the axis used this function, the following function can not be used.
- Absolute position detection (absolute pulse coder)
- Three-dimensional error correction

(6) If axial movement is made in the direction opposite to that of reference position return, the movement is reversed to the direction of reference position return after three or four reference marks have been detected. Steps 3 to 5 of the basic procedure for establishing a reference position are carried out to establish the reference position.

(7) Straightness compensation function
When the reference point establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by straightness compensation amount when the reference point of moving axis is established.

(8) The reference point establishment is not performed during synchronous control is activated.

(9) The reference point establishment is not performed during composite control is activated.

(10) The reference point establishment is not performed during superimposed control is activated.

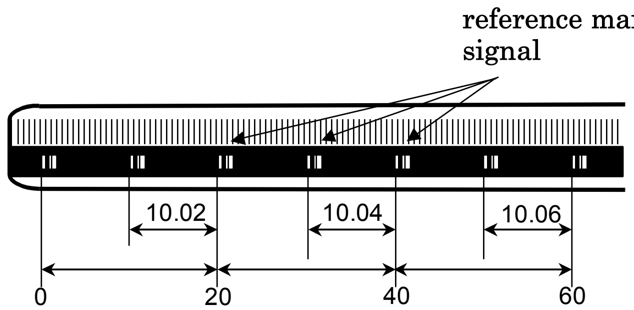
3.10 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)

Overview

By using High-resolution serial output circuit for the linear scale with distance-coded reference marks (serial), the CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position.
This function enables high-speed high-precision detection by using High-resolution serial output circuit.
It is available that using maximum stroke 30 meters length.

Explanation

The linear scale with distance-coded reference marks (serial) is combined the irregular reference marked linear scale with the High-resolution serial output circuit, it can detect the accurate position.

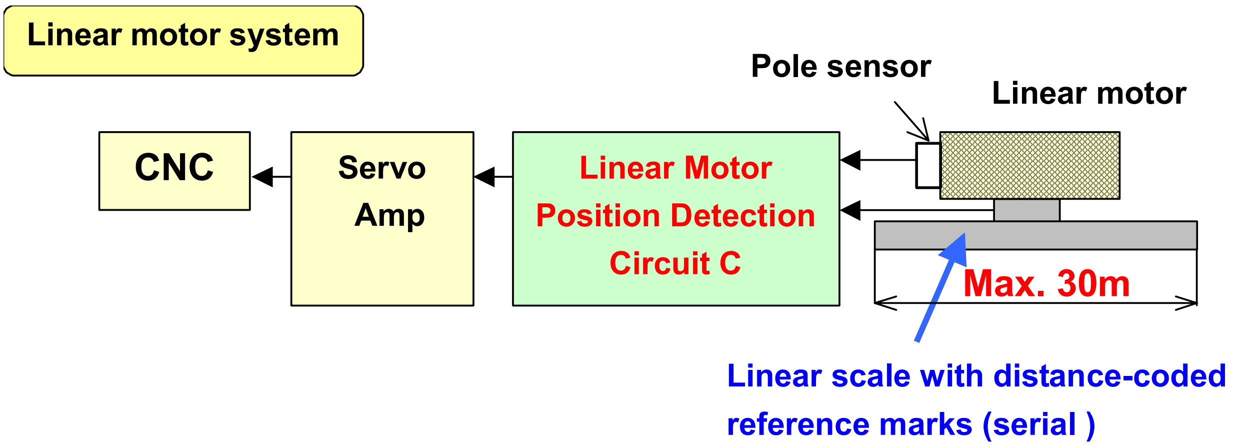


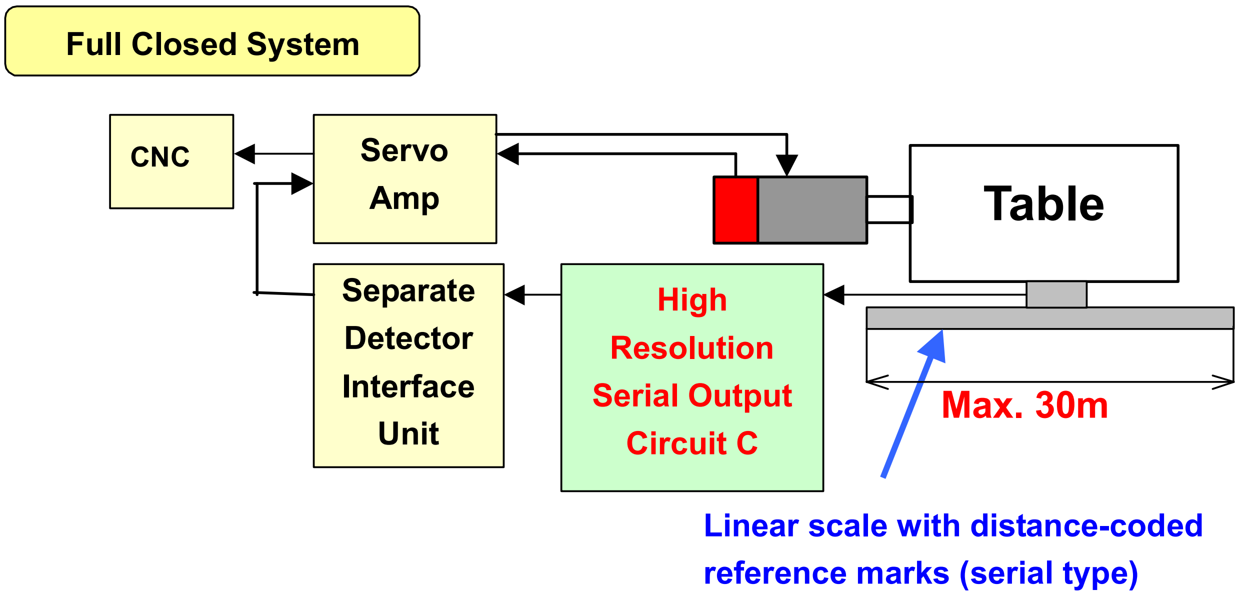
The CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position, because of the interval of each reference mark is different with regular interval.
It is not necessary that the axis is moved to the reference position for establishment of reference position.

This function enables high-speed high-precision detection by using High-resolution serial output circuit.
It is available that using maximum stroke 30 meters length.

- Connection

It is available under linear motor system and full closed system.

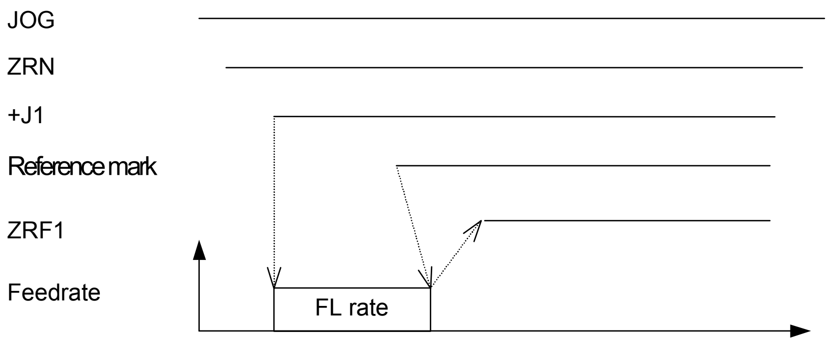




- Procedure for reference position establishment through manual operation

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set a direction selection signal(+J1,-J1,+J2,-J2,…) for a target axis.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter (No.1425) setting).
(4) When the absolute position of linear scale with distance-coded reference marks (serial) is detected, the axis stops. Then the absolute position of CNC is calculated and reference position establishment signal (ZRF1,ZRF2,ZRF3,…) turns to "1".

The timing chart for this procedures is given below.



- Procedure for reference position establishment through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (4) above are performed automatically.
After the reference position is established, the automatic reference position return is performed by setting of parameter RFS No.1818#0.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of the following operations is performed in steps (3) to (4), described above.

- Reset
- Setting the feed axis direction selection signal (+J1, -J1, +J2, -J2, etc.) to 0
- Setting the Servo off signals (SVF1, SVF2, etc.) to 1

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position
- Setting the Servo off signals (SVF1, SVF2, etc.) to 1

If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

- Establishing a reference position and moving to the reference position

By following operation, establishing a reference position and moving to the reference position is performed.

	Moving through manual operation in REF mode	**Moving through automatic operation by automatic reference position return (G28)**
The reference position is not established.	Establishing the reference position	Firstly, moving to the intermediate position, and establishing the reference position. Secondly, whether moving to the reference position or not is performed by setting parameter RFS No.1818#0.
The reference position is established.	Moving to the reference position	Whether moving to the intermediate position and the reference position or not is performed by setting parameter RF2 No.1818#1.

- Feed axis synchronization control

In case of using the axis synchronization control, please confirm the following items.

- When this function is used with axis synchronization control axes, the linear scale with distance-coded reference marks (serial) used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
- The master axis scale and the slave axis scale should be installed in parallel direction. (The zero positions should be faced the same direction.)
- To the parameters, which relate to this function (except No.1883, No.1884), the same value must be set for the master axis and for the slave axis.
- The linear scale with distance-coded reference marks (serial) should be applied for the master axis and the slave axis.
  If either of the master axis or the slave axis is not the linear scale with distance-coded reference marks (serial), alarm DS0018 occurs when reference position establishment is tried.
- During operating the establishment of reference position, the state of signal for selecting synchronized axis(SYNCn<Gn138> or SYNCJn<Gn140>) should be kept.

Procedure for reference position establishment by axis synchronization control is as follows.

- Both of axes (master axis and slave axis) are fed on the reference position return FL feedrate until distance coded scales of both axes detect the absolute position.
- Then absolute position of both axes are calculated and Reference Position Establishment Signals (ZRF1,ZRF2,...) turn to "1".

- Angular axis control

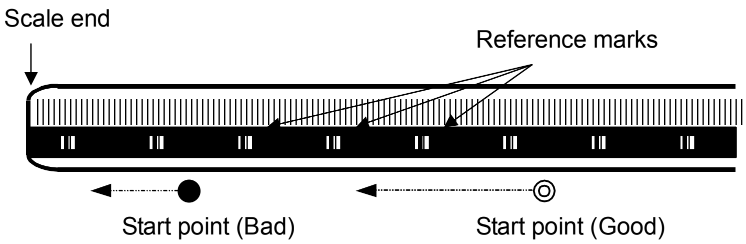
In case of using the angular axis control, please confirm the following items.

- It is necessary to use the linear scale with distance-coded reference marks (serial) for both the perpendicular axis and the angular axis.
  If not, the alarm DS0019 occurs when reference position establishment is commanded.
- When the reference point establishment of angular and perpendicular axes are tried, please set parameter 8200#2(AZR) to '0' and input signal G063#5(NOZAGC) to '0'.
  If not, the alarm DS0019 occurs when reference position establishment is commanded.
- When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the alarm DS0020 occurs.
- On angular axis control, if you use automatic setting of parameter No.1883,1884 on reference point establishment (parameter DATx No.1819#2=1), please establish reference point of perpendicular axis after reference point establishment and return of angular axis.

In manual reference position return, the perpendicular axis cannot be specified while the angular axis reference point is being established. The perpendicular axis, if specified, is ignored.

⚠ **CAUTION**

1 When the Linear scale with distance-coded reference marks (serial) is used, please set parameter SDCx No.1818#3 to 1.
2 And distance coded rotary encoder (serial type) is not available.
3 On the Linear scale with distance-coded reference marks (serial), the axis does not stop until three reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.
  And if establishment of reference position is failed, the establishment is retried. Then axis does not stop until still more three reference marks are detected. So please set the maximum move amount (detection unit : parameter No.14010) not to reach the scale end.



4 On flexible synchronization control mode, reference position can't be established.
5 Straightness compensation function
  When the reference point establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by straightness compensation amount when the reference point of moving axis is established.
6 It is not available to use this function and the temporary absolute coordinate setting together.

4 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.

This chapter explains the following types of automatic operation:

- MEMORY OPERATION
  Operation by executing a program registered in CNC memory
- MDI OPERATION
  Operation by executing a program entered from the MDI panel
- SUBPROGRAM CALL (M198)
  Function for calling and executing subprograms (files) registered in an external input/output device during memory operation
- MANUAL HANDLE INTERRUPTION
  Function for performing manual feed during movement executed by automatic operation
- MIRROR IMAGE
  Function for enabling mirror-image movement along an axis during automatic operation
- PROGRAM RESTART
  Restarting a program for automatic operation from an intermediate point
- TOOL RETRACT AND RECOVER
  Function for retracting the tool to a desired position

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.
When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.
When the RESET key on the MDI panel is pressed, automatic operation terminates and the reset state is entered.
For the multi-path control, the programs for the multiple paths can be executed simultaneously so the multiple paths can operate independently at the same time.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Memory operation

Procedure

1 Press the MEMORY mode selection switch.
2 Select a program from the registered programs. To do this, follow the steps below.
   2-1 Press PROG to display the program screen.
   2-2 Press address O .
   2-3 Enter a program number using the numeric keys.
   2-4 Press the [O SRH] soft key.
3 For the multi-path control, select the path to be operated with the path selection switch on the machine operator's panel.
4 Press the cycle start switch on the machine operator's panel. Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.
5 To stop or cancel memory operation midway through, follow the steps below.
   a. Stopping memory operation
      Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
      (i) When the machine was moving, feed operation decelerates and stops.
      (ii) When dwell was being performed, dwell is stopped.
      (iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
      When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.

b. Terminating memory operation

Press the RESET key on the MDI panel.

Automatic operation is terminated and the reset state is entered.
When a reset is applied during movement, movement decelerates then stops.

Explanation

- Memory operation

After memory operation is started, the following are executed:

(1) A one-block command is read from the specified program.
(2) The block command is decoded.
(3) The command execution is started.
(4) The command in the next block is read.
(5) Buffering is executed. That is, the command is decoded to allow immediate execution.
(6) Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.
(7) Hereafter, memory operation can be executed by repeating the steps (4) to.(6)

- Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

- The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).
- There are two keys to stop memory operation: The feed hold key and reset key.

- Program stop (M00)

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- Optional stop (M01)

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered.
In some machines, M30 returns control to the top of the program. For details, refer to the manual supplied by the machine tool builder.

- Feed hold

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

- Reset

Automatic operation can be stopped and the system can be made to the reset state by using RESET key on the MDI panel or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

- Optional block skip

When the optional block skip switch on the machine operator's panel is turned on, blocks containing a slash (/) are ignored.

- Cycle start for the multi-path control

For the multi-path control, a cycle start switch is provided for each path. This allows the operator to activate a single paths to operate them at the same time in memory operation or MDI operation. In general, select the path to be operated with the path selection switch on the machine operator's panel and then press the cycle start button to activate the selected path. (The procedure may vary with the machine tool builder. Refer to the appropriate manual issued by the machine tool builder.)

4.2 MDI OPERATION

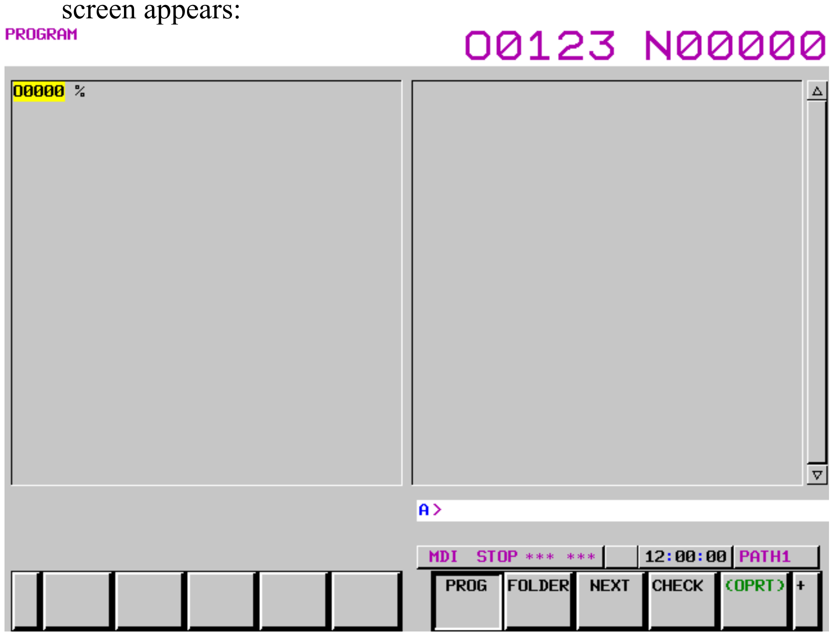
In the MDI mode, a program consisting of up to 255 characters can be created in the same format as normal programs and executed from the MDI panel.
MDI operation is used for simple test operations.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

MDI Operation

Procedure

1 Select the MDI mode.
For the multi-path control, select the path for which a program is to be created with the path selection switch. Create a separate program for each path.

2 Press the PROG key to select the program screen. The following screen appears:



MDI program screen

At this time, program number “O0000” is inserted automatically.

3 Prepare a program to be executed by an operation similar to normal program editing. M99 specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode.

4 To entirely erase a program created in MDI mode, use one of the following methods:
   a. Enter address O , then press the DELETE key.
   b. Alternatively, press the RESET key. In this case, set parameter MCL (No. 3203#7) to 1 in advance.

5 To execute a program, set the cursor on the head of the program. Push Cycle Start button on the operator's panel. By this action, the prepared program will start.
   (For the multi-path control, select the path to be operated with the path selection switch on the machine operator's panel beforehand.) When the program end (M02, M30) or ER(%) is executed, the prepared program will be automatically erased and the operation will end.
   By command of M99, control returns to the head of the prepared program.

6 To stop or terminate MDI operation in midway through, follow the steps below.
   a. Stopping MDI operation
      Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
      (i) When the machine was moving, feed operation decelerates and stops.
      (ii) When dwell was being performed, dwell is stopped.
      (iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
      When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.
   b. Terminating MDI operation
      Press the RESET key.
      Automatic operation is terminated and the reset state is entered.
      When a reset is applied during movement, movement decelerates then stops.

Explanation

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation, M30 does not return control to the beginning of the program (M99 performs this function).

- Erasing the program

Programs prepared in the MDI mode will be erased in the following cases:

- In MDI operation, if M02, M30 or ER(%) is executed.
- When bit 6 (MER) of parameter No. 3203 is set to 1, and the last block of the program is executed in single block operation

> **NOTE**
> In the two cases above, program erasure can be prevented by setting bit 6 (MKP) of parameter No. 3204 to 1.

- In MEM mode, if memory operation is performed.
- In EDIT mode, if any editing is performed.
- When the O and DELETE keys are pressed.
- Upon reset when parameter MCL (No. 3203#7) is set to 1

> **NOTE**
> Upon reset when the parameter MCL = 0, the cursor moves to the end of the program.

- Restart

If a program is not executed even once after the program is input, the program is executed from the beginning, regardless of where the cursor is placed. However, a program is executed starting at the beginning of the block where the cursor is place, if the program is stopped for a reason such as single block operation after restart of an MDI operation then is restarted after an editing operation.

> ⚠ **CAUTION**
> When an MDI program is restarted, the program is executed starting at the beginning of the block where the cursor is placed, regardless of the cursor position in the block.
> (Example)
> When the cursor is placed on G90
> :
> G91 X100.0 G90Y200.0 Z300.0 ;
> :
> The program is executed starting at the beginning (namely, G91) of this block. So, the tool moves by 100.0 along the X-axis in the incremental programming, and moves to 200.0 and 300.0 along the Y-axis and Z-axis, respectively, in the absolute programming.

- Editing a program during MDI operation

A program can be edited during MDI operation. By setting bit 5 (MIE) of parameter No. 3203 to 1, editing can be disabled. However, even when bit 5 (MIE) of parameter No. 3203 is set to 1, editing can be enabled by resetting the operation.

- Absolute/incremental command

When bit 4 (MAB) of parameter No. 3401 is set to 1, the absolute/incremental programming of MDI operation does not depend on G90/G91. In this case, the incremental programming is set when bit 5 (ABS) of parameter No. 3401 is set to 0, and the absolute programming is set when bit 5 (ABS) of parameter No. 3401 is set to 1.

Parameter MAB (No.3401#4)=0	**Parameter MAB (No.3401#4)=1**	
Absolute mode operation with G90 command, and incremental programming operation with G91 command	Parameter ABS (No.3401#5)=0	Parameter ABS (No.3401#5)=1
	Incremental mode operation at all times, independent of G90/G91 command	Absolute mode operation at all times, independent of G90/G91 command

> **NOTE**
> When G code system A is used on a lathe system, the parameters MAB and ABS are invalid.

Limitation

- Program registration

Programs created in MDI mode cannot be registered.

- Number of characters in a program

A created program can consist of up to 255 characters including "O0000" automatically inserted.

- Subprogram nesting

The subprogram call command (M98) can be described in a program created in MDI mode. That is, programs that are registered in memory through MDI operation can be called and executed. The level of subprogram call nesting is the same as in MEM operation.

- Macro call

When the custom macro function is enabled, a macro program can be created and executed even in the MDI mode. Moreover, a macro program can be called for execution.

NOTE

The GOTO statement, WHILE statement, and DO statement cannot be executed in a program created in the MDI mode. An alarm PS0377 is issued.
When a program including those statements is to be executed, register the program in the program memory then call the program for execution.

4.3 DNC OPERATION

By activating automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read in via reader/puncher interface, or remote buffer.

To use the DNC operation function, it is necessary to set the parameters related to the reader/punch interface, and remote buffer in advance.

The procedure described below is just an example. For actual operation, refer to the relevant manual of the machine tool builder.

DNC operation

Procedure

1 Select a program to be executed.

2 Press the REMOTE switch on the machine operator's panel to set RMT mode, then press the cycle start switch. The selected file is executed. For details of the use of the REMOTE switch, refer to the relevant manual supplied by the machine tool builder.

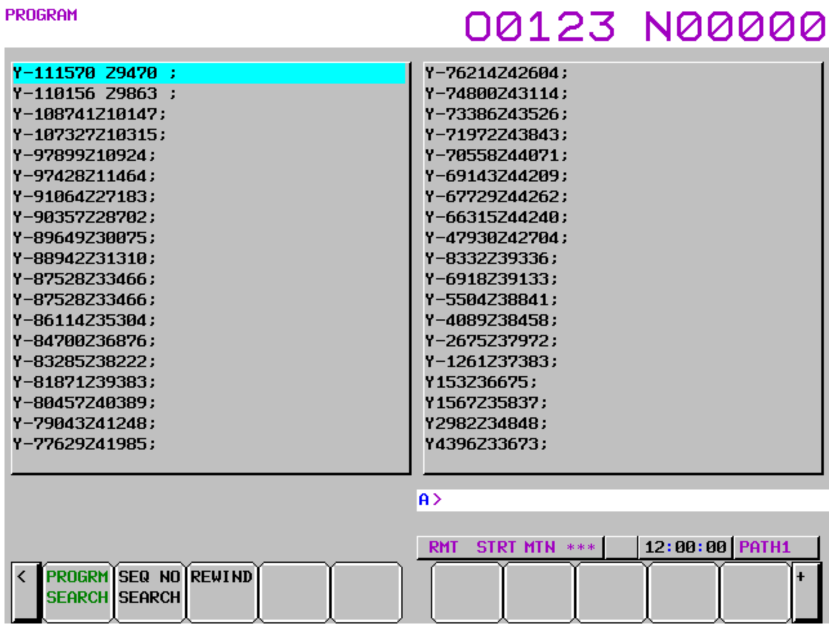


Fig. 4.3 (a) Program check screen

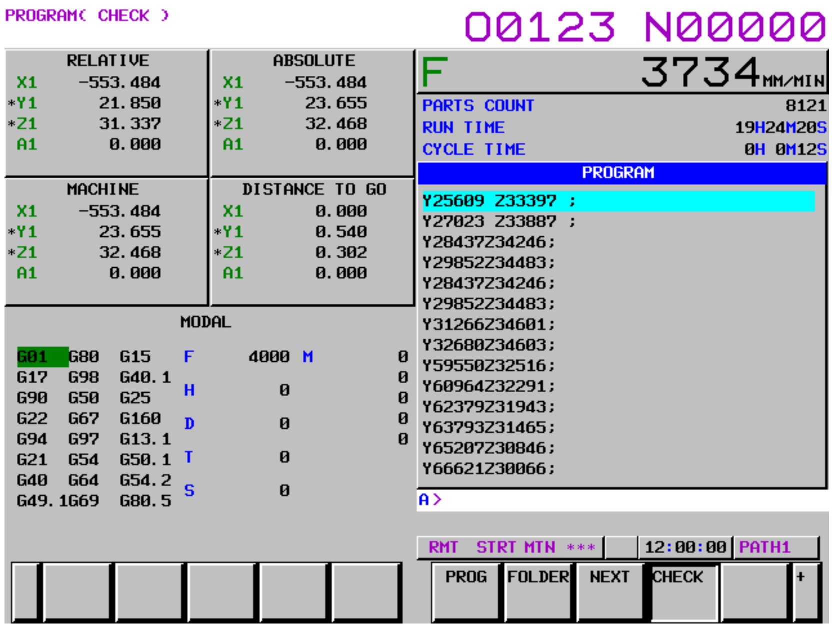


Fig. 4.3 (b) Program screen

During DNC operation, the program currently being executed is displayed on the program check screen and program screen.

Explanation

During DNC operation, subprograms and macro programs stored in memory can be called.

Limitation

- M198 (command for calling a program from within an external input/output unit)

In DNC operation, M198 cannot be executed. If M198 is executed, alarm PS0210 is issued.

- Custom macro

In DNC operation, custom macros can be specified, but no repeat instruction and branch instruction can be programmed. If a repeat instruction or branch instruction is executed, alarm PS0123 is issued.

- M99

For returning from a subprogram or macro program to the calling program during DNC operation, the specification of a return command (M99P::::) with a sequence number specified is not allowed.

4.4 EXTERNAL SUBPROGRAM CALL (M198)

During memory operation, you can call and execute a subprogram registered in an external device (such as a Memory Card, Handy File, or Data Server) connected to the CNC.

Format

M198 Pxxxxxxxx Lyyyyyyyy ;

↑ ↑

Pxxxxxxxx : Program number (or file number)

Lyyyyyyyy : Number of repetitive calls

When address L is omitted, the number of repetitive calls is assumed to be 1.

FS16 compatible command format

(The following command format is valid only when a 4-digit program number is used.)

M198 P<u>xxxx</u> <u>yyyy</u> ;

↑ ↑

<u>xxxx</u> : Number of repetitive calls

<u>yyyy</u> : Program number (or file number)

When the number of repetitive calls is omitted, it is assumed to be 1.

Explanation

M code M198 specifies an external subprogram call. You can also call an external subprogram using an M code set in parameter No. 6030. (When an M code other than M198 is set as an M code for calling an external subprogram, M198 is executed as a normal M code.)

Specify a program number (file number) registered in an external device at address P. If the specified program number (file number) is not registered in the connected external device, an alarm (PS1079) is issued.

Example)

M198 P0123 L3;

This command specifies that the subprogram having external subprogram number O0123 is to be called three times repeatedly.
The subprogram is called from the main program and executed as follows:

Main program	1 2 3	Sub program
N0010 ... ;		0123 ... ;
N0020 ... ;		N1020 ... ;
N0030 M198 P0123 L3 ;		N1030 ... ;
N0040 ... ;		N1040 ... ;
N0050 ;		N0050 ... ;
N0060 ... ;		N1060 ... M99 ;

- Program number call

You can also specify a subprogram call with its program number instead of the file number by the setting of bit 2 (SBP) of parameter No. 3404.

NOTE

1 An external subprogram call can be specified only during program operation in the MEM mode and cannot be specified in the MDI mode.

2 An external subprogram call is available for the following external devices:

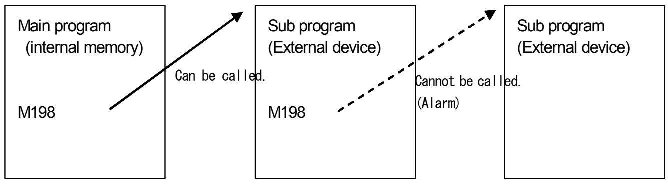
External device name	Program number call	File number call
Handy File	○	○
FLOPPY CASSTTE	○	○
Memory Card	○	×
Data Server	○	×

(○: Available ×: Unavailable)

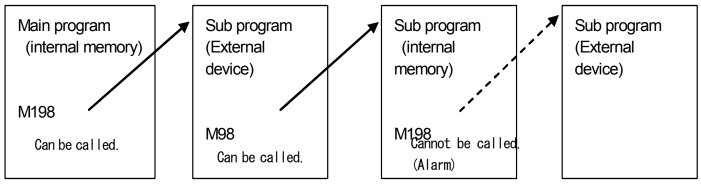
3 To perform a subprogram call using a Memory Card as the external device, set bit 7 (MNC) of parameter No. 138 to 1 and I/O channel (parameter No. 0020) to 4.
A program number call is always enabled regardless of the setting of bit 2 (SBP) of parameter No. 3404.

NOTE

4 An external device subprogram call cannot be performed from a subprogram called using another external device subprogram call. (An alarm (PS1080) is issued.)



5 A subprogram registered in internal memory can be called from a subprogram called using an external device subprogram call. From the called subprogram in internal memory, another external device subprogram call cannot be performed. (An alarm (PS1080) is issued.)



6 A call using the external device subprogram call function is counted as one level of subprogram nesting.

7 In a multipath system, a subprogram call cannot be performed simultaneously from multiple paths.

4.5 MANUAL HANDLE INTERRUPTION

By rotating the manual pulse generator in the automatic operation mode (manual data input, DNC operation, or memory operation) or in the memory editing mode, handle feed can be superimposed on movement by automatic operation. A handle interruption axis is selected using the manual handle interruption axis selection signal.
The minimum unit of travel distance per scale division is the least input increment. One of four types of magnifiers selected with MP1 and MP2 <G019#4 and #5> can be applied. With bit 3 (HIT) of parameter No. 7103, the minimum unit of travel distance can be further 10 times greater. A handle feed magnifier is selected using the manual handle feed amount selection signal. (See "MANUAL HANDLE FEED".)

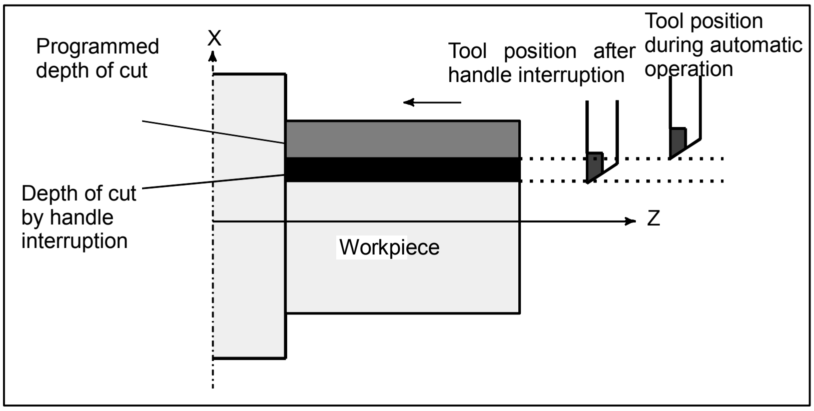


Fig. 4.5(a) Manual handle interruption

> ⚠ **WARNING**
> The travel distance per scale division by manual handle interruption is the least input increment as with manual handle feed. With a machine of metric input and inch output, for example, the travel distance per 254 scale divisions is 0.01 inch. With a machine of inch input and metric output, the travel distance per 100 scale divisions is 0.254 mm.

Explanation

- Interruption operation

1 When the handle interruption axis selection signal for a handle interruption axis is set to 1 in the automatic operation mode (manual data input, DNC operation, or memory operation) or in the memory editing mode, manual handle interruption can be performed by rotating the handle of the manual pulse generator.

> **NOTE**
> Even when the feedrate override signal sets 0%, manual handle interruption can be accepted.

2 For the method of selecting a manual handle interruption axis, refer to the relevant manual of the machine tool builder.

3 The feedrate during manual handle interruption is the sum of feedrate used for automatic operation and the feedrate used for movement by manual handle interruption. However, the feedrate during manual handle interruption is controlled so that it does not exceed the maximum allowable cutting feedrate for the axis.

> **Example**
> Suppose that the maximum allowable cutting feedrate for an axis is 5 m/min, and that a movement is made in the + direction at 2 m/min along the axis. In this case, manual handle interruption can be accepted even when the manual pulse generator is rotated up to a speed equivalent to 3 m/min. Manual handle interruption by rotation in one direction can be accepted even when the manual pulse generator is rotated to a speed equivalent to 7 m/min.
> If the manual pulse generator is rotated to a speed beyond the upper limits, those pulses from the manual pulse generator that correspond to the excess are lost, resulting in a mismatch between the scale mark of the manual pulse generator and the actually interrupted travel distance.

4 For a magnifier for manual handle interruption, refer to the relevant manual of the machine tool builder.

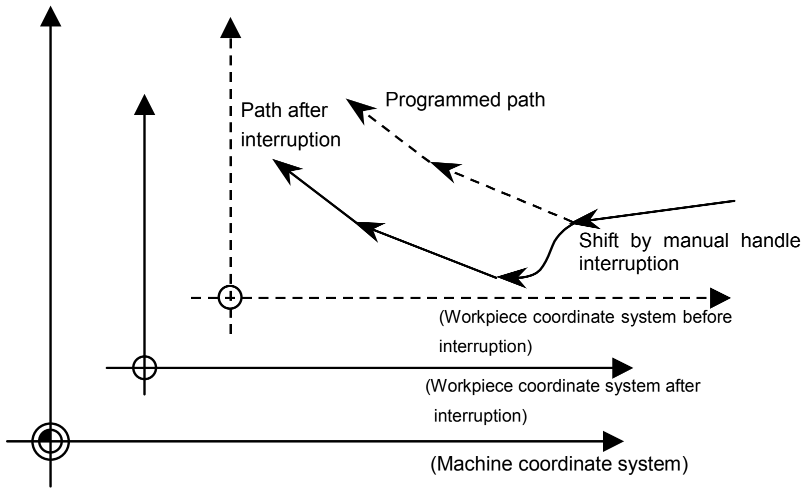
5 If the travel direction is reversed as a result of manual handle interruption, backlash compensation is performed. Pitch error compensation is performed for the position after interruption.

6 In manual handle interruption, only acceleration/deceleration for cutting feed is enabled. By setting bit 0 (MNJ) of parameter No. 1606 to 1, acceleration/deceleration for both of cutting feed and jog feed can be applied to manual handle interruption.

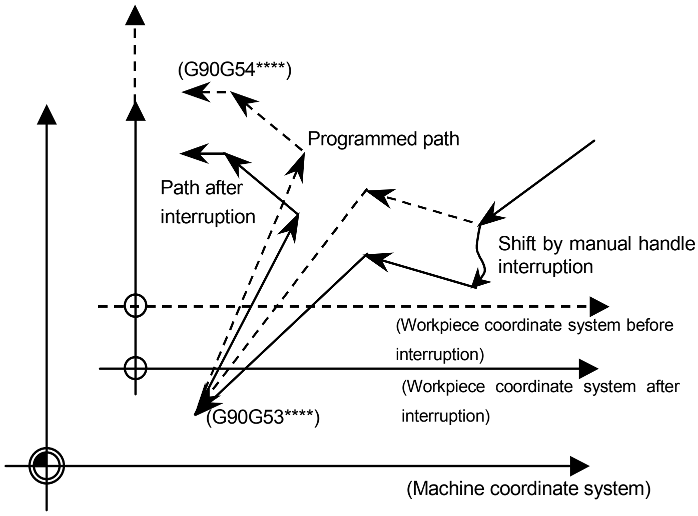
- Manual handle interruption and coordinate system

1 The amount of manual handle interruption shifts the workpiece coordinate systems and the local coordinate system. So, the machine moves, but the coordinates in the workpiece coordinate systems and the local coordinate system remain unchanged.
Regardless of which coordinate system is selected, all workpiece coordinate systems and the local coordinate system shift by the same amount.

- Absolute coordinates
  → Remain unchanged by handle interruption.
- Relative coordinates
  → Change by the amount of handle interruption.
- Machine coordinates
  → Change by the amount of handle interruption.



2 Even when manual handle interruption is performed, the machine coordinate system remains unchanged. The absolute command (G53) in the machine coordinate system is not affected by manual handle interruption.

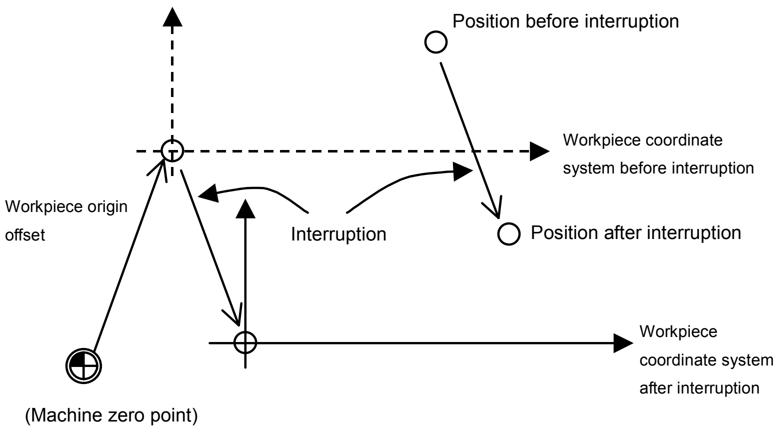


3 In automatic reference position return (G28), the end point (reference position) is not affected by manual handle interruption. However, the midpoint is in the workpiece coordinate system, so that the position shifted by the amount of interruption becomes the midpoint.

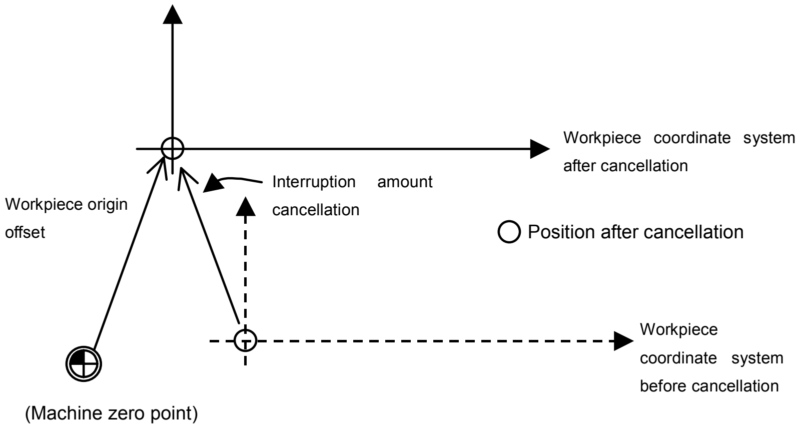
- Cancellation of the amount of interruption

Operation by which the workpiece coordinate system shifted by manual handle interruption from the machine coordinate system is returned to the original workpiece coordinate system is referred to cancellation of the amount of interruption.
When the amount of interruption is canceled, the workpiece coordinate system is shifted by the amount of manual handle interruption, and the amount of interruption is reflected in the absolute coordinates.

Interruption shifts the workpiece coordinate system from the machine coordinate system.



By cancellation, the workpiece coordinate system returns to the state present before handle interruption.



In the following cases, the amount of interruption is canceled:

- When a reset is made (when bit 1 (RTH) of parameter No. 7103 is set to 1)
- When emergency stop state is canceled (when bit 1 (RTH) of parameter No. 7103 is set to 1)
- When a manual reference position return operation is performed (when G28 is specified before a reference position is established)
- When a reference position is set without dogs
- When the workpiece coordinate system is preset

NOTE

When the amount of interruption is cleared using soft keys, only the indication of the amount of interruption becomes 0, and the workpiece coordinate system remains unchanged.

- Relation with other functions

The following table indicates the relation between other functions and the movement by handle interruption.

Table 4.5(a) Relation between other functions and the movement by handle interruption

Signals	Relation
Machine lock	Machine lock is effective. When machine lock is on, no movement is made due to handle interruption.
Interlock	Interlock is effective. When interlock is on, no movement is made due to handle interruption.
Mirror image	Mirror image is not effective. Interrupt functions on the plus direction by plus direction command, even if this signal turns on.

- Position display

The following table shows the relation between various position display data and the movement by handle interruption.

Table4.5(b) relation between various position display data and the movement by handle interruption

Signals	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Relative coordinates are changed by the travel distance specified by handle interruption.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

- Travel distance display

Press the function key POS , then press the chapter selection soft key [HNDL].

The move amount by the handle interruption is displayed. The following 4 kinds of data are displayed concurrently.

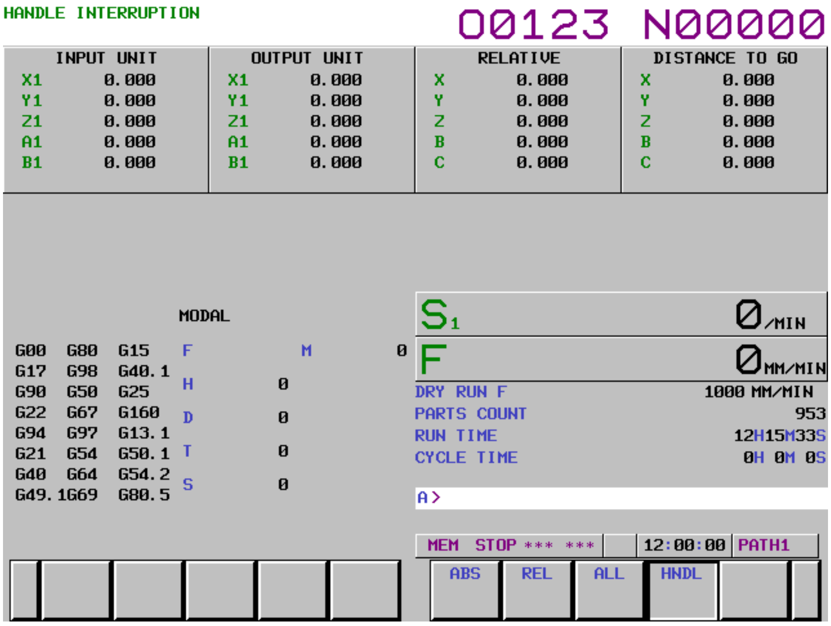


Fig. 4.5 (b)

(a) INPUT UNIT:
Handle interruption move amount in input unit system
Indicates the travel distance specified by handle interruption according to the least input increment.

(b) OUTPUT UNIT :
Handle interruption move amount in output unit system
Indicates the travel distance specified by handle interruption according to the least command increment.

(c) RELATIVE:
Position in relative coordinate system
These values have no effect on the travel distance specified by handle interruption.

(d) DISTANCE TO GO:
The remaining travel distance in the current block has no effect on the travel distance specified by handle interruption.

The handle interruption move amount is cleared when the manual reference position return ends every axis.

- Display for five-axis systems or better

Systems having five or more axes provide the same display as the overall position display. See III-12.1.3.

4.6 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. To use this function, set the mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the MDI panel.

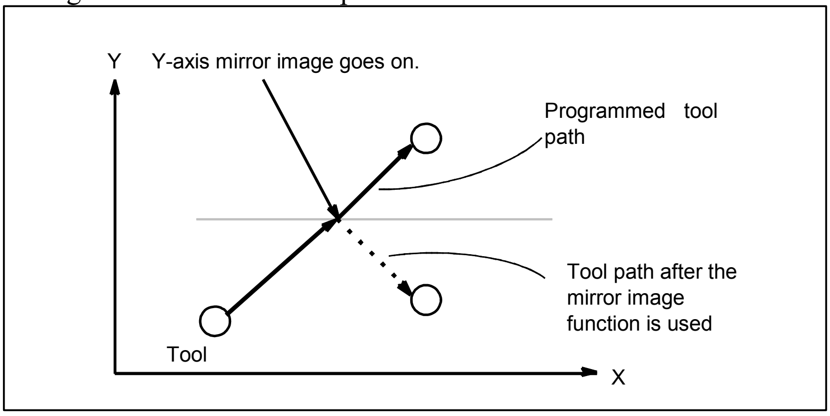


Fig. 4.6 (a) Mirror image

Procedure for mirror image

Procedure

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

1 Press the single block switch to stop automatic operation.
When the mirror image function is used from the beginning of operation, this step is omitted.

2 Press the mirror image switch for the target axis on the machine operator's panel.
Alternatively, turn on the mirror image setting by following the steps below:
2-1 Set the MDI mode.
2-2 Press the OFFSET SETTING function key.

2-3 Press the [SETING] soft key for chapter selection to display the setting screen.

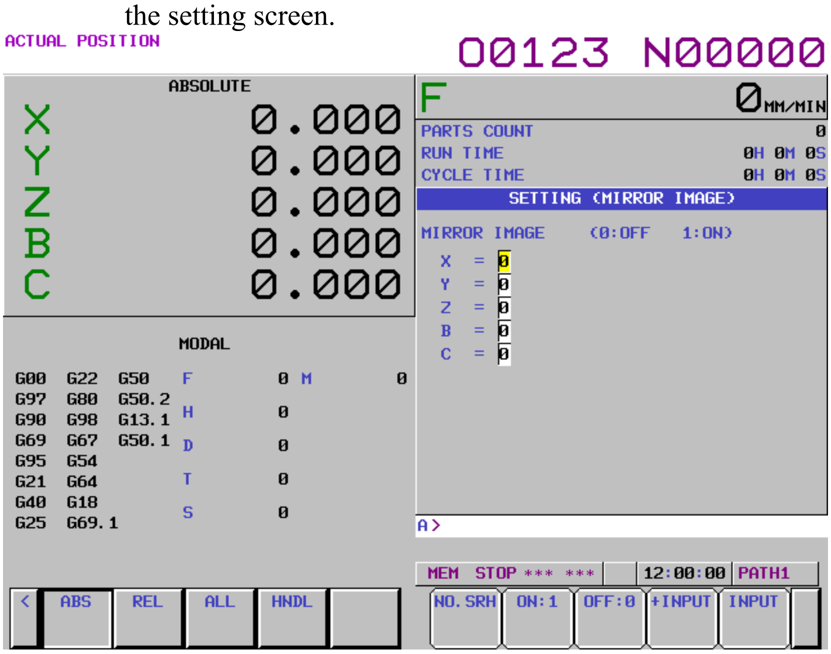


Fig. 4.6 (b) Setting screen

2-4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (memory mode or MDI mode), then press the cycle start button to start automatic operation.

Explanation

- The mirror image function can also be turned on and off by setting parameter MIRx (No. 0012 #0) to 1 or 0.
- For the mirror image switches, refer to the manual supplied by the machine tool builder.

Limitation

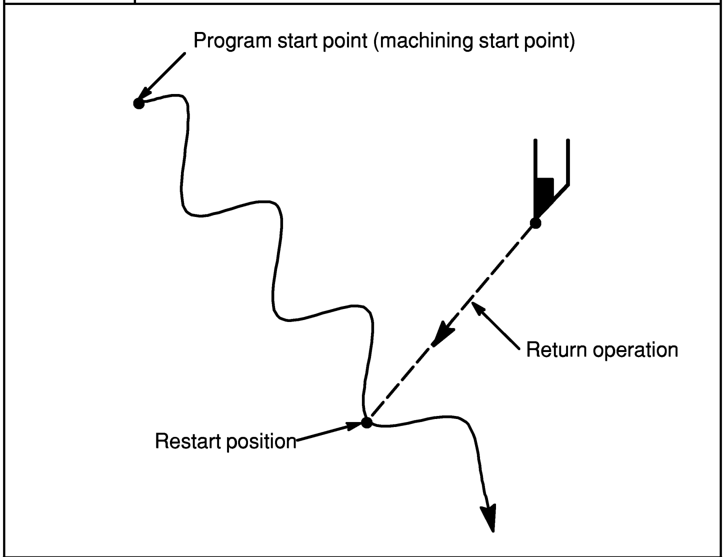
The direction of movement during manual operation, the direction of movement from an intermediate point to the reference position during automatic reference position return (G28), the direction of approach during single direction positioning (G60), and the shift direction in a boring cycle (G76, G87) cannot be reserved.

4.7 PROGRAM RESTART

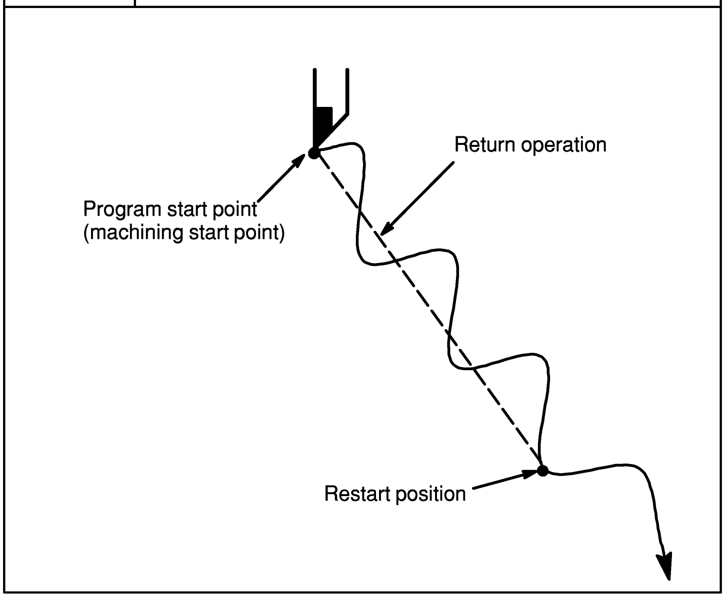
This function specifies Sequence No. of a block to be restarted when a tool is broken down or when it is desired to restart machining operation after a day off, and restarts the machining operation from that block. It can also be used as a high-speed program check function.

There are two restart methods: the P-type method and Q-type method.

P TYPE	Operation can be restarted anywhere. This restart method is used when operation is stopped because of a broken tool.



Q TYPE	Before operation can be restarted, the machine must be moved to the programmed start point (machining start point)



Procedure for program restart by specifying a sequence number

Procedure 1

[P TYPE]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

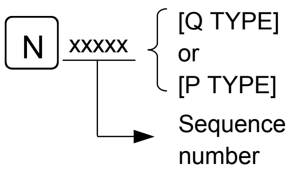
[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
3 If necessary, modify the offset amount. (Go to step 2.)

Procedure 2

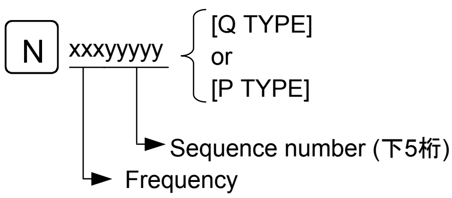
[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.
2 Press PROG key to display the desired program.
3 Find the program head. Press RESET key.
4 Enter the sequence number of the block to be restarted, then press the [P TYPE] or [Q TYPE] soft key.



If the same sequence number appears more than once, the location of the target block must be specified. Specify a frequency and a sequence number.

5 The sequence number is searched for, and the program restart screen appears on the LCD display.



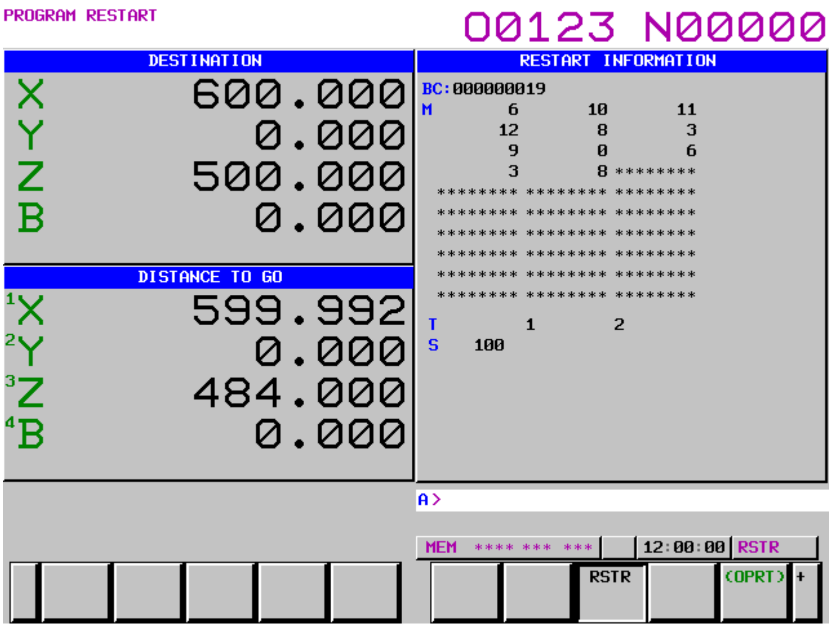


Fig. 4.7 (a) Program restart screen

DESTINATION shows the position at which machining is to restart.

DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.

The coordinates and amount of travel for restarting the program can be displayed for up to five axes. If your system supports six or more axes, pressing the [RSTR] soft key again displays the data for the sixth and subsequent axes.

M : Up to 35 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of the display.
With 15” LCD/MDI panel or 10.4” LCD/MDI panel :
Up to 35 M codes
With 9.5” LCD/MDI panel : Up to 14 M codes

T : Two most recently specified T codes

S : Most recently specified S code

B : Most recently specified B code

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the

machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter settings (No. 7310). Machining is then restarted.

Procedure for program restart by specifying a block Number

Procedure 1
[P TYPE]

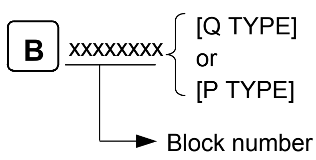
1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
3 If necessary, modify the offset amount. (Go to step 2.)

Procedure 2
[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.
2 Press PROG key to display the desired program.
3 Find the program head. Press RESET key.
4 Enter the number of the block to be restarted then press the [P TYPE] or [Q TYPE] soft key. The block number cannot exceed eight digits.



5 The block number is searched for, and the program restart screen appears on the LCD display.

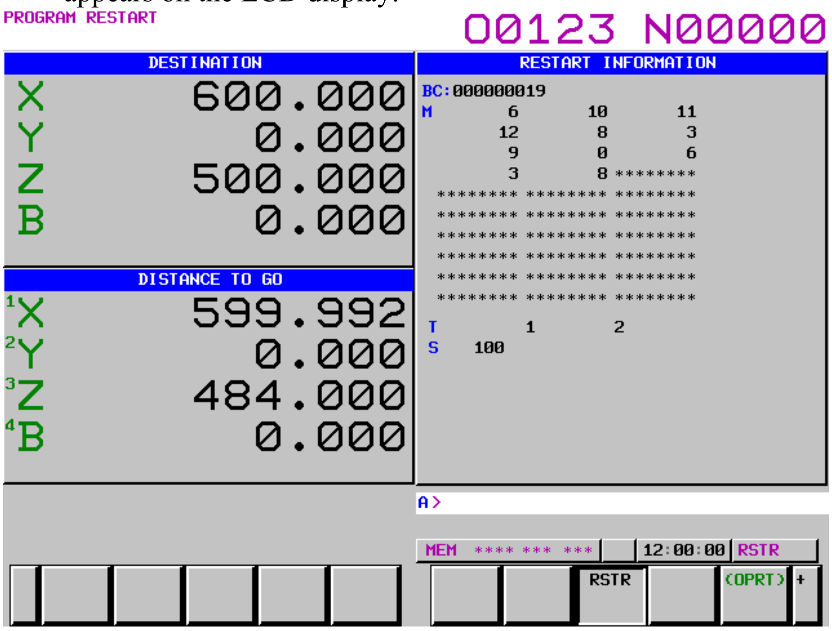


Fig. 4.7 (b) Program restart screen

DESTINATION shows the position at which machining is to restart.
DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.
The coordinates and amount of travel for restarting the program can be displayed for up to five axes. If your system supports six or more axes, pressing the [RSTR] soft key again displays the data for the sixth and subsequent axes.

M : Up to 35 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of the display.
With 15” LCD/MDI panel or 10.4” LCD/MDI panel :
Up to 35 M codes
With 9.5” LCD/MDI panel : Up to 14 M codes

T : Two most recently specified T codes

S : Most recently specified S code

B : Most recently specified B code
Codes are displayed in the order in which they are specified.
All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter settings (No. 7310). Machining is then restarted.

Outputting the M, S, T, and B codes for program restart

After the block to be restarted is searched for, you can perform the following operations:

1 Before the tool is moved to the machining restart position

<1> The most recently specified M, S, T, and B codes can automatically be output to the PMC.
The most recently specified S code is output as the maximum spindle speed when the S code is specified in the block containing G92 or as the specified spindle speed in other cases. As the most recently specified S code, only one S code is displayed on the program restart screen regardless of whether the S code is specified in the block containing G92.

<2> While the block to be restarted is being searched for, all sampled M codes and most recently specified S, T, and B codes can automatically be output to the PMC. Up to 35 M codes can be sampled. If the number of sampled M codes exceeds 35, the 35 most recently specified M codes are output to the PMC.

Switch between operations <1> and <2> using bit 6 (MOA) of parameter No. 7300.

2 Before the tool reaches the machining restart position
On the program restart screen, you can specify M, S, T, and B codes from the MDI panel in the MEM or RMT mode without changing the mode.

Outputting the most recently specified M, S, T, and B codes

When bit 7 (MOP) of parameter No. 7300 is set to 1, pressing the cycle start switch after searching for the block to be restarted automatically outputs the most recently specified M, S, T, and B codes to the PMC before the tool is moved to the machining restart position.
In the single block stop status, after the most recently specified M, S, T, and B codes are output, pressing the cycle start switch again moves the tool to the machining restart position.

Outputting all M codes and most recently specified S, T, and B codes

When bit 6 (MOA) of parameter No. 7300 is set to 1, pressing the cycle start switch after searching for the block to be restarted automatically outputs all M codes and most recently specified S, T, and B codes to the PMC before the tool is moved to the machining restart position.
(Example)

When M10, M11, M12, M13, M14, T0101, S1000, and B10 are sampled, a program is executed in the format shown below before the tool is moved to the machining restart position:

```
M10 T0101 S1000 B10 ;
M11 ;
M12 ;
M13 ;
M14 ;
```

Outputting M, S, T, and B codes on the program restart screen

When bit 7 (MOP) of parameter No. 7300 is set to 1, you can specify M, S, T, and B codes from the MDI panel in the MEM or RMT mode without changing the mode after searching for the block to be restarted until the tool reaches the machining restart position.

Procedure

1 When the block to be restarted is searched for using the program restart function, the program restart screen appears. When bit 7 (MOP) of parameter No. 7300 is set to 1, operation soft keys [OVERSTORE], [ERASE], and [INPUT] are displayed.

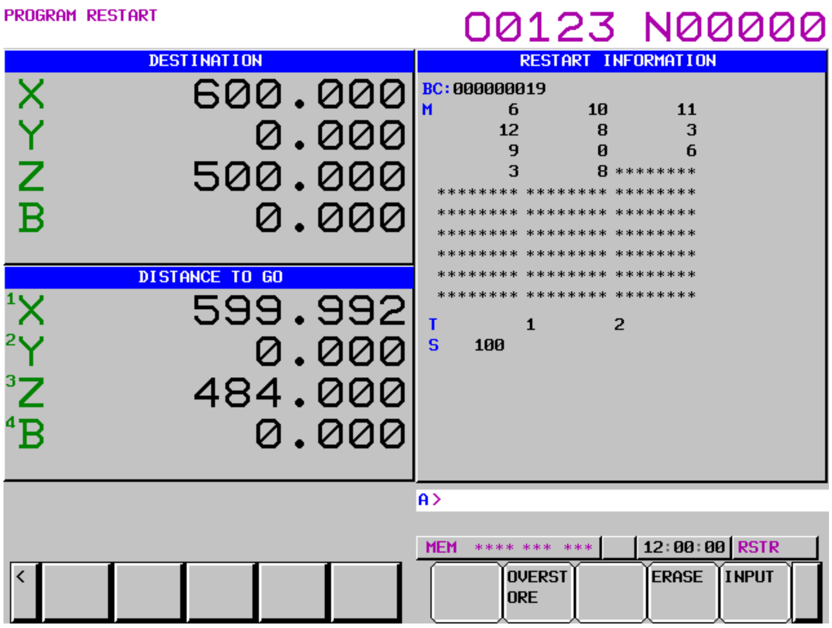


Fig. 4.7 (c) Program restart screen (outputting M, S, T, and B codes)

2 Before the tool reaches the machining restart position, pressing soft key [OVERSTORE] selects the over store mode. In the over store mode, data can be entered in the M, S, T, and B fields displayed in the (OVERSTORE) section.
To select the over store mode while the tool is moving to the machining restart position, hold restart operation by feed hold and press soft key [OVERSTORE].
Enter M, S, T, and B codes to be output in the (OVERSTORE) section from the MDI panel.
(Example)
To enter M10, S1000, T101, and B20 in the (OVERSTORE) section:
<1> Enter M 1 0 from the MDI panel.
<2> Press the [INPUT] key.
You can also enter the S, T, and B codes by performing steps <1> and <2>.

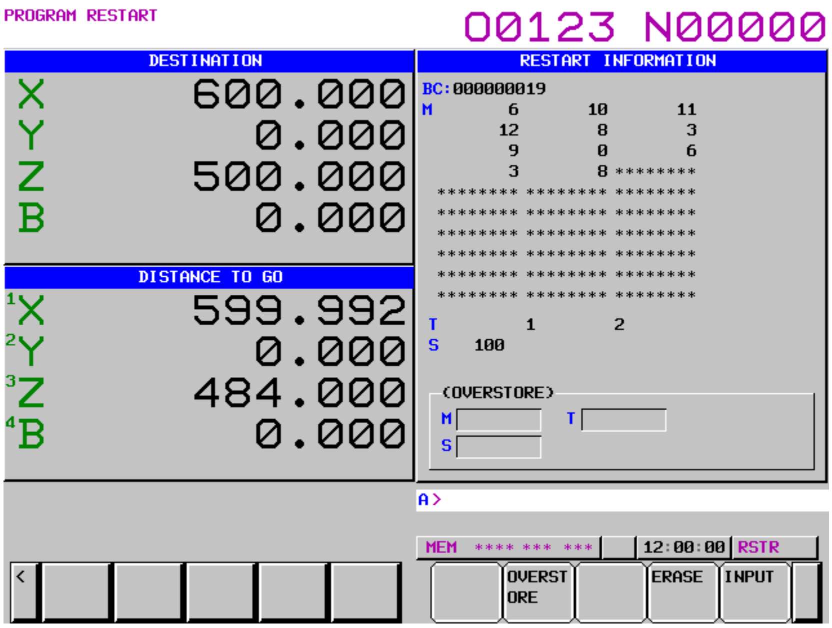


Fig. 4.7 (d) Program restart screen when M, S, T, and B codes are output

3 When values have been entered in the (OVERSTORE) section, pressing the cycle start switch outputs each code in the (OVERSTORE) section. The values in the (OVERSTORE) section are cleared.
4 To clear the values entered in the (OVERSTORE) section as M, S, T, and B codes, press soft key [ERASE]. All entered values are cleared.
5 Pressing soft key [OVERSTORE] again in the over store mode cancels the mode. Pressing the reset key also cancels the over store mode.
6 To continue with restart operation, cancel the over store mode and press the cycle start switch.

⚠ **CAUTION**

1 The M, S, T, and B codes specified in the over store mode are not displayed on the program restart screen.
2 In the over store mode, changing the operation mode to other than the MEM or RMT mode does not cancel the over store mode. In this case, no values can be entered in the (OVERSTORE) section.

Explanation

- Block number

When the CNC is stopped, the number of executed blocks is displayed on the program screen or program restart screen. The operator can specify the number of the block from which the program is to be restarted, by referencing the number displayed on the LCD. The displayed number indicates the number of the block that was executed most recently. For example, to restart the program from the block at which execution stopped, specify the displayed number, plus one.
The number of blocks is counted from the start of machining, assuming one NC line of a CNC program to be one block.

(Example 1)

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G01 X100. F100 ;	3
G03 X01 -50. F50 ;	4
M30 ;	5

(Example 2)

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G90 G00 Z100. ;	3
G81 X100. Y0. Z120. R-80. F50. ;	4
#1=#1+1 ;	4
#2=#2+1 ;	4
#3=#3+1 ;	4
G00 X0 Z0 ;	5
M30 ;	6

Macro statements are not counted as blocks.

- Storing / clearing the block number

The block number is held in memory while no power is supplied. The number can be cleared by cycle start in the reset state.

- Block number when a program is halted or stopped

The program screen usually displays the number of the block currently being executed. When the execution of a block is completed, the CNC is reset, or the program is executed in single-block stop mode, the program screen displays the number of the program that was executed most recently. When a CNC program is halted or stopped by feed hold, reset, or single-block stop, the following block numbers are displayed:

Feed hold : Block being executed
Reset : Block executed most recently
Single-block stop : Block executed most recently

For example, when the CNC is reset during the execution of block 10, the displayed block number changes from 10 to 9.

- MDI intervention

When MDI intervention is performed while the program is stopped by single-block stop, the CNC commands used for intervention are not counted as a block.

- Block number exceeding eight digits

When the block number displayed on the program screen exceeds eight digits, the block number is reset to 0 and counting continues.

Limitation

- P type Restart

In the following conditions, P type restart cannot be performed:

- Automatic operation has not been performed since power-on.
- Automatic operation has not been performed since emergency stop was released.
- Automatic operation has not been performed since the coordinate system was changed or shifted (change of the external workpiece origin offset value).

The block that can be restored properly by P type program restart is a block for which coordinate system setup or change was performed most recently before machining was interrupted.

- Restart block

The block where the program is to restart is not necessarily be the block at which the program was interrupted. You can restart the program from any block. For P-type restart, however, the block where the program is to restart must use the same coordinate system as when program execution was interrupted.

- Single block

When the single-block operation is enabled at the time of a movement to the restart point, a single-block stop occurs each time an axis operation takes place. In this case, no MDI operation is allowed.

- Manual intervention

During movement to the restart point, manual intervention is allowed for an axis for which a return operation has not yet been performed. However, manual operations do not cause any movement along axes for which a return operation has already been completed.

- MDI

When the search operation has ended, no move command can be specified by MDI before axis movement.

- Reset

Do not perform a reset operation during the time from the start of the search operation of the restart sequence until machining is restarted.
If a reset operation is performed, the restart steps must be performed again from the beginning.

- Feed hold

If a feed hold operation is performed during the search, the restart steps must be performed again from the beginning.

- Manual absolute

Every manual operation must be performed with the manual absolute mode turned on regardless of whether the manual operation is performed before or after machining.

- Reference position return

Unless an absolute position detector (absolute pulse coder) is provided, be sure to perform reference position return after power-up, then perform restart operation.

- Program restart switch

When the program restart switch is on, pressing the cycle start switch does not start operation.

- Blocks specifying a macro statement, macro call, and subprogram call

Blocks specifying a macro statement, macro call, and subprogram call are not searched for even when they have a sequence number. In such a case, search for a block previously preceding such a block.

- Interruption type custom macro

During movement to the machining restart point at a dry run feedrate, no interruption type custom macro can be started. If an interruption type custom macro is started, alarm DS024 is issued.

- Commands that prevent program restart

Program restart cannot be performed for blocks placed in the following modes:

- Cs contouring control
- Polygon turning (G50.2)
- Threading (G32,G33), Circular threading (G35,G36), Threadingcycle (G92), Multiple repetitive threading cycle (G76)
- Polar coordinate interpolation (G12.1)
- Balance cutting (G68)
- Rigid tapping

If any of the following commands is included between the beginning of a program and the block where the program is to restart, program restart cannot be performed:

- Hypothetical axis interpolation
- Workpiece coordinate system preset (G92.1,G50.3)
- Commands for enabling and disabling synchronous/mixture control and superimposed control
- Commands for enabling and disabling axis synchronous control

- M, S, and T commands not usable in over store mode

The M, S, and T functions listed below, unlike the other M, S, and T functions, have special meanings within the CNC. These M, S, and T commands cannot be specified from the over store screen. To specify these commands, cancel the over store mode, and execute them in MDI operation.

Example:

- Spindle positioning
- Rigid tapping
- Tool management

> ⚠ **WARNING**
>
> As a rule, the tool cannot be returned to a correct position under the following conditions.
> Special care must be taken in the following cases since none of them cause an alarm:
>
> - Manual operation is performed when the manual absolute mode is OFF.
> - Manual operation is performed when the machine is locked.
> - When the mirror image is used. However, P type return is possible for a block that switched between ON and OFF most recently or a subsequent block. In this case, the mirror image signal status present when the program was interrupted must be maintained.
> - When no coordinate system is set up at the beginning of a program in which main commands are executed in the incremental mode.
> - When manual operation is performed in the course of axis movement for returning operation.
> - When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
> - When program restart is specified in the machine lock state, then the machine lock is canceled.
> - When program restart specified for an intermediate block for a multiple repetitive canned cycle
> - In general, when a coordinate system is set up, changed, or shifted after the search operation ends, the tool cannot be returned to a correct position.

⚠ **CAUTION**

Keep the following in mind when restarting a program including macro variables.

- Common variable
  When the program is restarted, the previous values are inherited as common variables without being preset automatically. Before restarting the program, initialize the appropriate variables to the original values used at start of the previous automatic operation.
- DI/DO
  At restart of the program, DI can be read by a system variable, but DO cannot be output.
- Clock
  When the program is being restarted, the clock time can be obtained by a system variable, but the time cannot be preset.
- Tool offset and workpiece origin offset
  When the program is being restarted, the offset can be read by a system variable, but change of the offset is allowed only for the Q type.

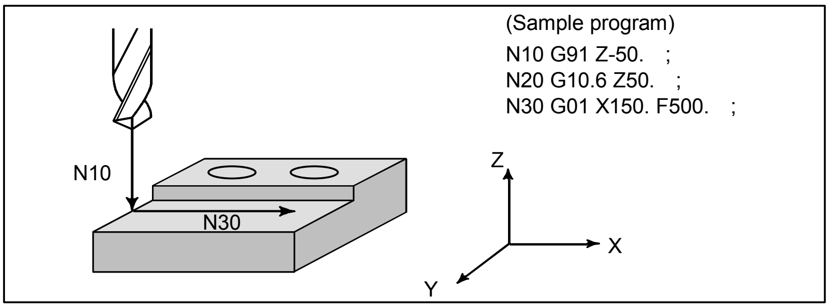
4.8 TOOL RETRACT AND RECOVER

The tool can be retracted from a workpiece to replace the tool, if damaged during machining, or to check the status of machining. Then, the tool can be returned to restart machining efficiently.

Procedure for tool retract and recover

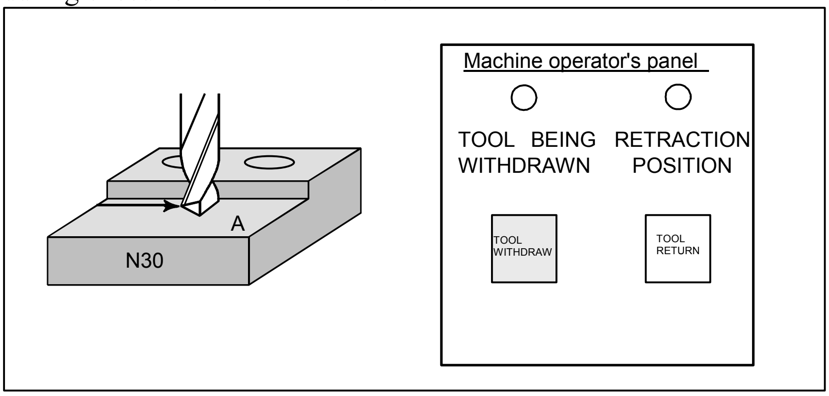
Procedure1 - Programming

Specify a retraction axis and distance in command G10.6IP_ beforehand.In the sample program below, the N20 block specifies that the Z-axis is the retraction axis and the retraction distance is to be 50 mm.

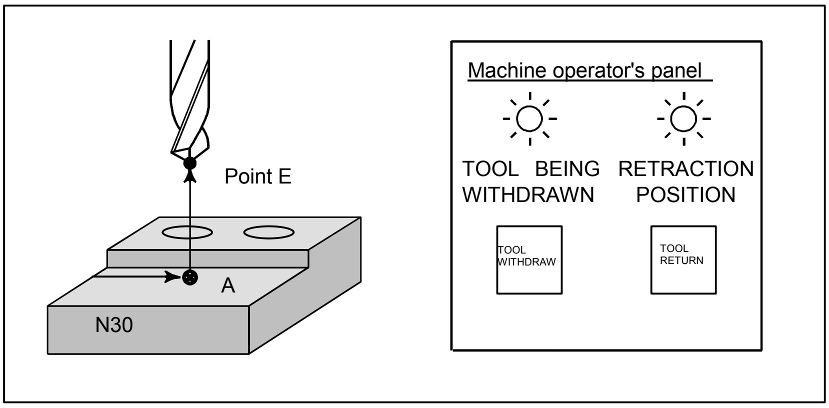


Procedure 2 - Retract

Suppose that the TOOL WITHDRAW switch on the machine operator's panel is turned on when the tool is positioned at point A during execution of the N30 block.



Next, the tool withdrawal mode is set and the TOOL BEING WITHDRAWN LED goes on. At this time, automatic operation is temporarily halted. The tool is then retracted by the programmed distance. If point A is the end point of the block, retraction is performed after automatic operation is stopped. Retraction is based on linear interpolation. The dry run feedrate is used for retraction. Upon completion of retraction, the RETRACT POSITION LED on the operator's panel goes on.



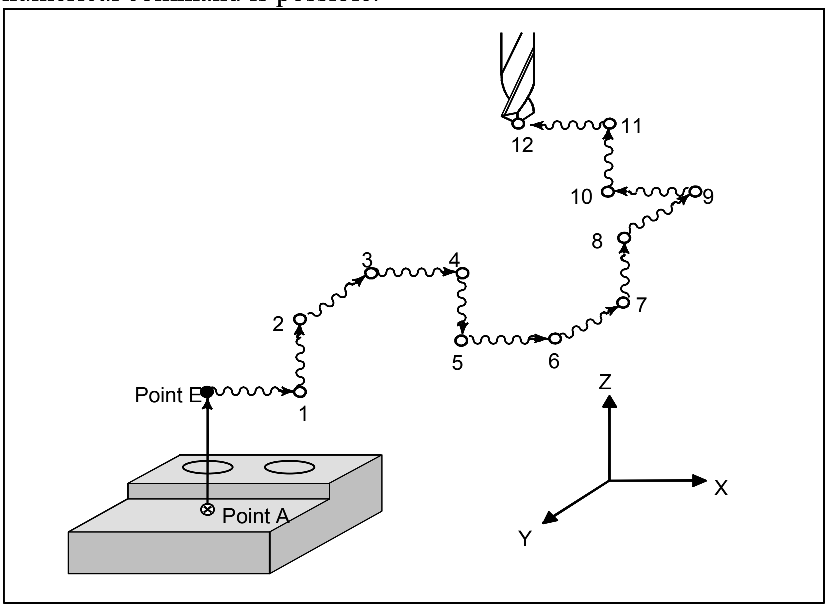
During retraction, the LCD screen displays PTRR and STRT.

MEM STRT MTN *** 12:00:00 PTRR

- PTRR blinks in the field for indicating states such as the program editing status.
- STRT is displayed in the automatic operation status field.
- MTN is displayed in the field for indicating status such as movement along an axis.

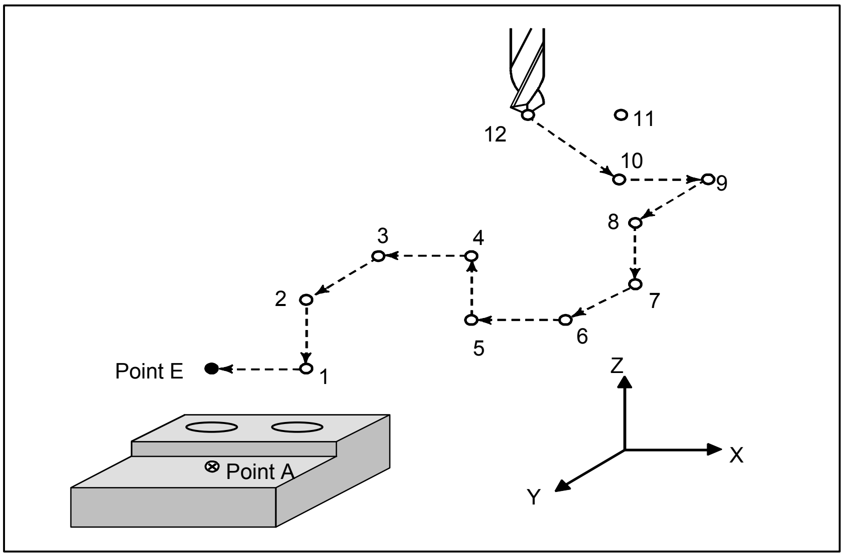
Procedure 3 - Withdrawal

Set the manual operation mode, then withdraw the tool. For manual operation, either jog feed incremental feed, handle feed, or manual numerical command is possible.



Procedure 4 - Return

After withdrawing the tool and any additional operation such as replacing the tool, move the tool back to the previous retraction position. To return the tool to the retraction position, return the mode to automatic operation mode, then turn the TOOL RETURN switch on the operator's panel on then off again. The tool returns to the retraction position at the dry run feedrate, regardless of whether the dry run switch is on or off.
When the tool has returned to the retraction position, the RETRACTION POSITION LED comes on.

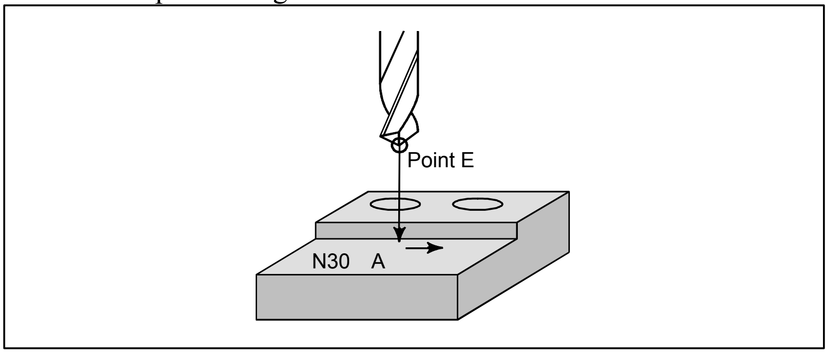


During return operation, the LCD screen displays PTRR and MSTR.

- PTRR blinks in the field for indicating states such as program editing status.
- MSTR is displayed in the automatic operation status field.
- MTN is displayed in the field for indicating states such as movement along an axis.

Procedure 5 - Repositioning

While the tool is at the retraction position (point E in the figure below) and the RETRACTION POSITION LED is on, press the cycle start switch. The tool is then repositioned at the point where retraction was started (i.e. where the TOOL WITHDRAW switch was turned on). Repositioning is based on linear interpolation. The dry run feedrate is used for repositioning.



Upon completion of repositioning, the tool withdraw mode is cancelled, and the TOOL BEING WITHDRAWN LED goes off and restart N30.

Limitation

1. If the origin, presetting, workpiece origin offset value (or external workpiece origin offset value), or workpiece coordinate shift amount (for a lathe system) is changed after the retraction position is specified with G10.6 in the absolute mode, the change is not reflected in the retraction position. After such changes are made or the workpiece origin offset value (or external workpiece origin offset value) or workpiece coordinate shift amount (for a lathe system) is changed, respectively the retraction position with G10.6.
2. When retracting the tool manually in the tool withdrawal mode, do not use the machine lock, mirror-image, or scaling function.

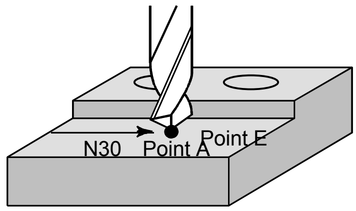
> ⚠ **WARNING**
> The retraction axis and retraction distance specified in G10.6 must be changed in an appropriate block according to the figure being machined. Be very careful when specifying the retraction distance; an incorrect retraction distance may damage the workpiece, machine, or tool.

4.8.1 Retract

Explanation

- When no retraction distance is specified

If no retraction distance or direction required for retraction are specified, retraction is not performed when the TOOL WITHDRAW switch on the operator's panel is turned on. Instead, the block being executed in automatic operation is interrupted (automatic operation is held or stopped). In this state, the tool can be withdrawn and returned.



- Retraction from the automatic operation hold or stop state

When the single block switch is turned on during automatic operation, or the TOOL WITHDRAW switch is turned on after the automatic operation hold or stop state is set by feed hold: Retraction is performed, then the automatic operation hold or stop state is set again.

- Stopping retraction

During retraction, feed hold operation is ignored. However, reset operation is enabled (retraction is stopped at reset). When an alarm is issued during retraction, the retraction is stopped immediately.

- Repositioning immediately after retraction

After retraction is completed, tool repositioning can be started without performing the withdraw and return operations.

4.8.2 Withdrawal

Explanation

- Axis selection

To move the tool along an axis, select the corresponding axis selection signal. Never specify axis selection signals for two or more axes at a time.

- Path memorization

When the tool is moved in manual operation along an axis, the control unit memorizes up to ten paths of movements. If the tool is stopped after being moved along a selected axis and is then moved along another selected axis, the position where this switch takes place is memorized. After ten paths have been memorized, the control unit does not memorize any additional switching points.

- Reset

Upon reset, memorized position data is lost and the tool withdraw mode is cancelled.

NOTE

If an attempt is made to move the tool simultaneously along two axes using the manual numeric command in the tool withdrawal mode, an alarm (PS0015) is issued.

4.8.3 Return

Explanation

- Return path

When there are more than ten return paths, the tool first moves to the tenth position, then to the ninth position, then to the eighth position, and so forth until the retraction position is reached.

- Single block

The single block switch is enabled during return operation. If the single block switch is turned off, continuous return operation is performed. If the single block switch is turned off, the tool stops at each memorized position. In this case, return operation can be resumed by turning the TOOL RETURN switch on then off again.

- Interruption of return operation

When an alarm is issued during return operation, return operation stops.

- Feed hold

The feed hold function is enabled during return operation.

4.8.4 Repositioning

Explanation

- Feed hold

The feed hold function is disabled during repositioning.

- Operation after completion of repositioning

The operation after completion of repositioning depends on the automatic operation state present when the TOOL WITHDRAW switch is turned on.

1 When automatic operation is being started
After completion of repositioning, the interrupted execution of the block is resumed.
2 When automatic operation is held or stopped
After completion of repositioning, the tool stops once at the repositioned point, then the original automatic operation hold or stop state is set. When the cycle start switch is pressed, automatic operation is resumed.

4.8.5 Tool Retract and Return for Threading

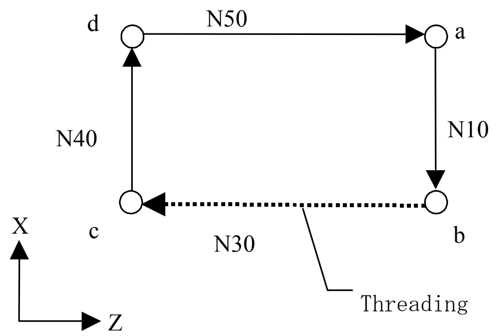
Explanation

- Differences between ordinary tool retract and return and tool retract and return for threading

1. During retraction, chamfering is performed between the specified retraction axis and threading axis.
2. After retraction, one block that does not specify threading is executed and the tool stops.
3. When the major axis for threading is specified as the retraction axis, retraction is not performed by turning the TOOL WITHDRAW switch on. In this case, after a block that does not specify threading is executed, an alarm (PS0429) is issued and the tool stops.
4. As repositioning, the tool is returned to the position specified in the first block that does not specify threading.

- Operation procedure

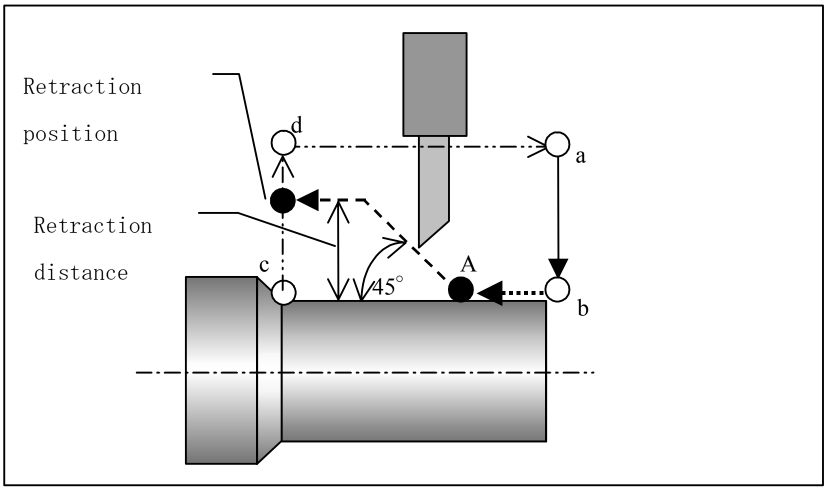
1. Specify a retraction axis and retraction distance in command "G10.6IP- -;".




```
O1234
G90G0X0Z0 ;
S150 M03 ;
N10 G91 G00 X-50. ;
N20 G10.6 X40.0 ;
N30 G33 Z-100. F2.0 ;
N40 G00 X50. ;
N50 Z100. ;
M02;

Retraction axis: X
Retraction distance: 40.0
```

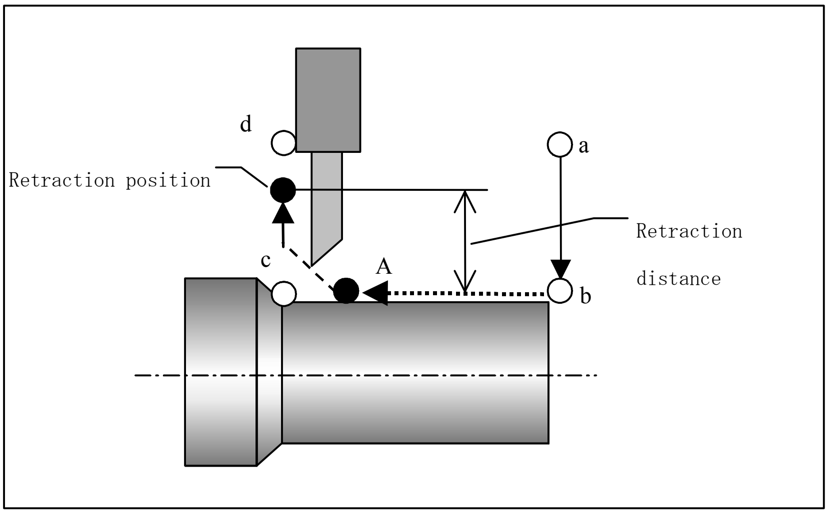
2. Turn the TOOL WITHDRAW switch during the execution of a threading command block.
3. The tool withdrawal mode is set and retraction is performed. Chamfering at 45 degrees is performed between the retraction axis and major axis for threading using the retraction distance as the chamfering amount during retraction.
   Details of retraction differ depending on whether the remaining travel distance for the threading command is smaller than the retraction distance when the TOOL WITHDRAW switch is turned on as follows:

(1) When remaining travel distance for threading ≥ retraction distance



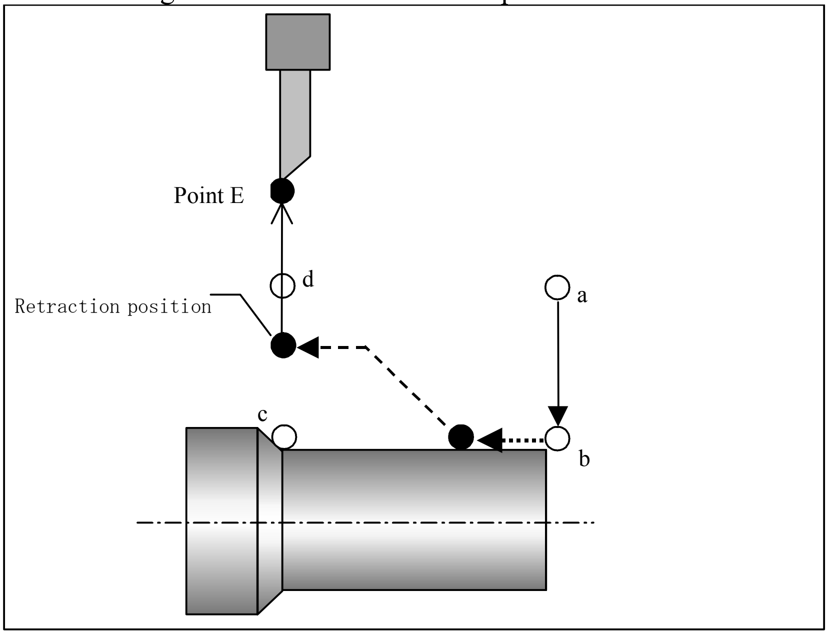
When the position where 45-degree chamfering by the retraction distance ends does not exceed the threading end position (c), the tool moves to the threading end position after the termination of chamfering.

(2) When the remaining travel distance for threading < retraction distance



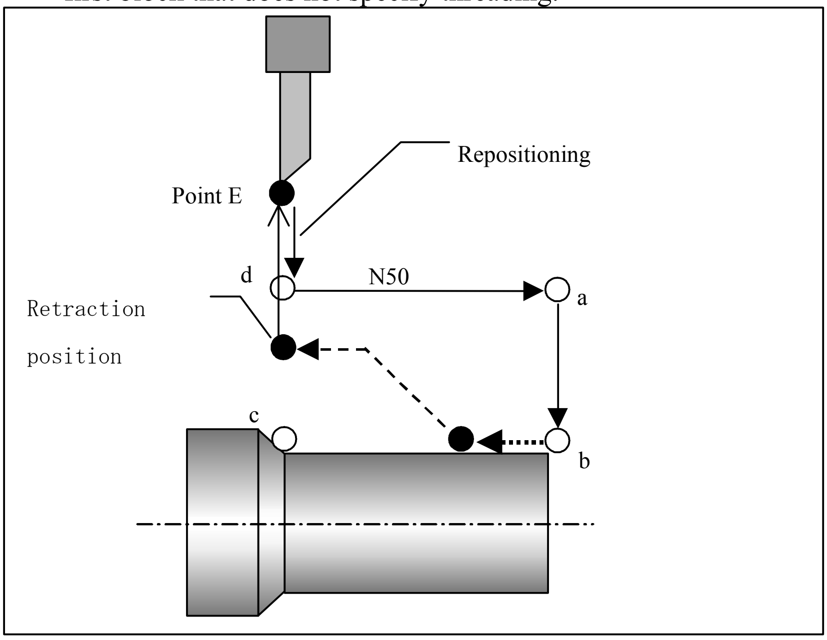
When the position where 45-degree chamfering by the retraction distance ends exceeds the threading end position (c), the tool moves to the retraction position along the retraction axis after it reaches the threading end position.

4 After retraction is completed, the next block that does not specify threading is executed and the tool stops.



In this example, "X50.0" is specified in the first block that does not specify threading in the incremental mode, the tool moves to point E and stops.
If the major axis for threading is specified as the retraction axis, the block that does not specify threading is executed without performing retraction, an alarm (PS0429) is issued, and the tool stops.

5 As repositioning, the tool returns to the position specified in the first block that does not specify threading.



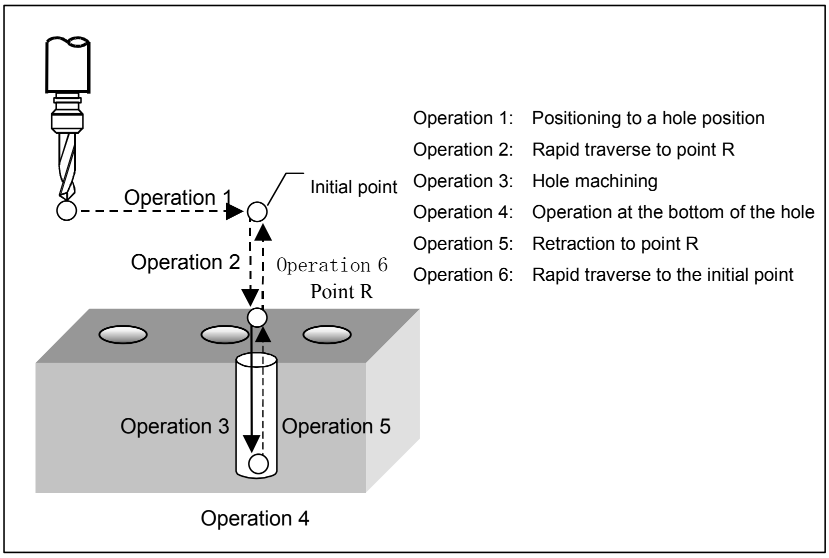
In this example, the repositioning position is point d. Automatic operation after repositioning starts at the N50 block.

4.8.6 Operation Procedure for a Canned Cycle for Drilling

Explanation

- Retract

When the TOOL WITHDRAW switch is turned on during a canned cycle for drilling (abbreviated as a canned cycle below), retraction is performed depending on the cycle operation being executed at that time.



1 During operation 1, the tool moves by the retraction distance specified in G10.6 in the same way as for ordinary retraction.
2 During operation 2, the tool stops operation 2, moves to the initial point, and stops.
3 During operation 3, the tool stops operation 3, executes cycle operations 4, 5, and 6 from that position, and stops at the initial point.
4 During operation 4, 5, or 6, the tool continues the operation and stops at the initial point.
When the TOOL WITHDRAW switch is turned on during operation 2 to 6, the tool does not move according to the retraction specified in G10.6. After the TOOL WITHDRAW switch is turned on and the tool moves to the initial point, however, the tool withdrawal mode is set.
When the second or subsequent canned cycle is being executed and the TOOL WITHDRAW switch is turned on during operation 2 to 6, the retraction position differs depending on G98 (return to initial level) or G99 (return to point R level).
   - G98 (return to initial level): The tool moves to the initial level.
   - G99 (return to point R level):The tool moves to the point R level.
5 During operation 2 to 6, the tool also moves to the initial point and stops when the TOOL WITHDRAW switch is turned on without the G10.6 command specified.

- Repositioning

When the tool is at the retraction position and the cycle start switch is pressed, repositioning is performed for the canned cycle.

1 Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 1
After the completion of repositioning, automatic operation is resumed in the same way as for ordinary repositioning.

2 Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 2
The canned cycle is reexecuted from operation 2.

3 Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 3
The canned cycle is reexecuted from operation 2.

4 Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 4, 5, or 6
The canned cycle is reexecuted for the same hole position from operation 2.

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the tool, use machine lock.
There are two types of machine lock: all-axis machine lock, which stops the movement along all axes, and specified-axis machine lock, which stops the movement along specified axes only. In addition, auxiliary function lock, which disables M, S, T, and B (2nd auxiliary function) commands, is available for checking a program together with machine lock.

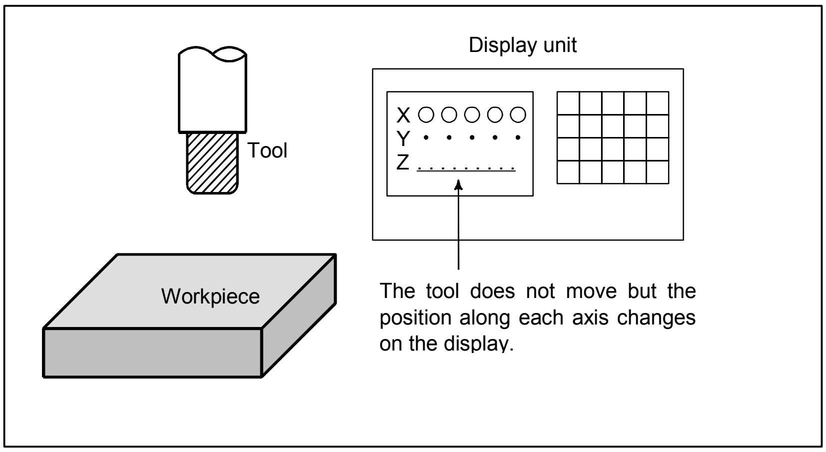


Fig. 5.1 (a) Machine lock

Machine lock and auxiliary function lock

Procedure
- Machine Lock

Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving.
Some machines have a machine lock switch for each axis. On such machines, press the machine lock switches for the axes along which the tool is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.

> ⚠ **WARNING**
> The positional relationship between the workpiece coordinates and machine coordinates may differ before and after automatic operation using machine lock. In such a case, specify the workpiece coordinate system by using a coordinate setting command or by performing manual reference position return.

- Auxiliary function lock

Press the auxiliary function lock switch on the operator's panel. M, S, T, and B codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Limitation

- M, S, T, B command by only machine lock

M, S, T and B commands are executed in the machine lock state.

- Reference position return under machine lock

When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

- M codes not locked by auxiliary function lock

M00, M01, M02, M30, M98, and M99 commands are executed even in the auxiliary function lock state. M codes for calling a subprogram (parameters No. 6071 to 6079) and those for calling a custom macro (parameters No. 6080 to 6089) are also executed.

5.2 FEEDRATE OVERRIDE

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial. This feature is used to check a program.
For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

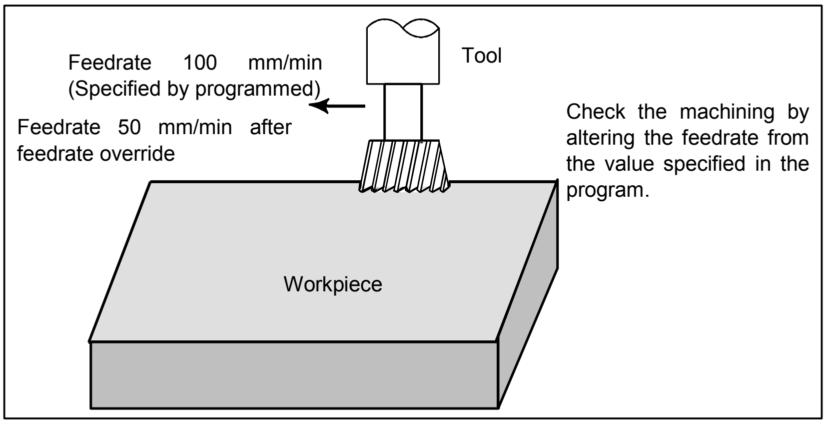


Fig. 5.2 (a) Feedrate override

Feedrate override

Procedure

Set the feedrate override dial to the desired percentage (%) on the machine operator's panel, before or during automatic operation.
On some machines, the same dial is used for the feedrate override dial and jog feedrate dial. Refer to the appropriate manual provided by the machine tool builder for feedrate override.

Limitation

- Override range

The override that can be specified ranges from 0 to 254%. For individual machines, the range depends on the specifications of the machine tool builder.

- Override during thread

During the threading process, the override setting is ignored; it is always regarded as 100% during the process.

5.3 RAPID TRAVERSE OVERRIDE

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter (No. 1421).

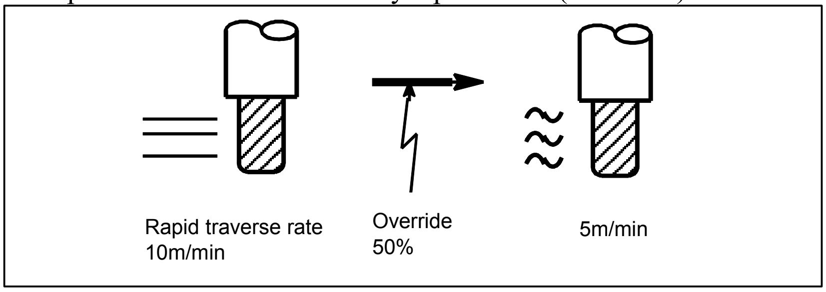


Fig. 5.3 (a) Rapid traverse override

Rapid traverse override

Procedure

Select one of the four feedrates with the rapid traverse override switch during rapid traverse. Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.

Explanation

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

(1) Rapid traverse by G00
(2) Rapid traverse during a canned cycle
(3) Rapid traverse in G27, G28, G29, G30, G53
(4) Manual rapid traverse
(5) Rapid traverse of manual reference position return

5.4 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

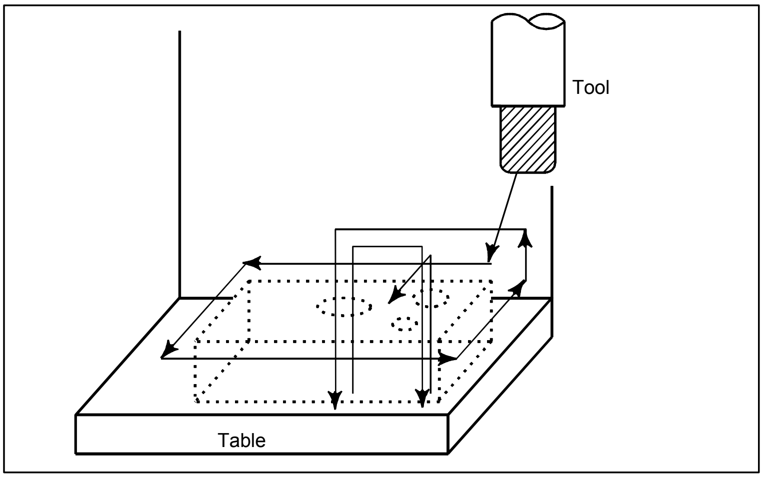


Fig. 5.4 (a) Dry run

Dry run

Procedure

Press the dry run switch on the machine operator's panel during automatic operation.
The tool moves at the feedrate specified in a parameter. The rapid traverse switch can also be used for changing the feedrate.
Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

- Dry run feedrate

The dry run feedrate changes as shown in the table below according to the rapid traverse switch and parameters.

Table 5.4 (a)

Rapid traverse switch	Program command	
	Rapid traverse	Cutting feed
ON	Rapid traverse rate	Dry run feedrate × Jvmax(*2)
OFF	Dry run feedrate × JV, or rapid traverse rate (*1)	Dry run feedrate × JV(*2)

Max. cutting feedrate .... Setting by parameter No.1430
Rapid traverse rate .... Setting by parameter No.1420
Dry run feedrate .... Setting by parameter No.1410

(*1) Dry run feedrate × JV when parameter RDR (No. 1401#6) is 1. Rapid traverse rate when parameter RDR is 0.
JV Jog feedrate override

(*2) Clamped to the maximum cutting feedrate
Jvmax Maximum value of jog feedrate override

5.5 SINGLE BLOCK

Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.

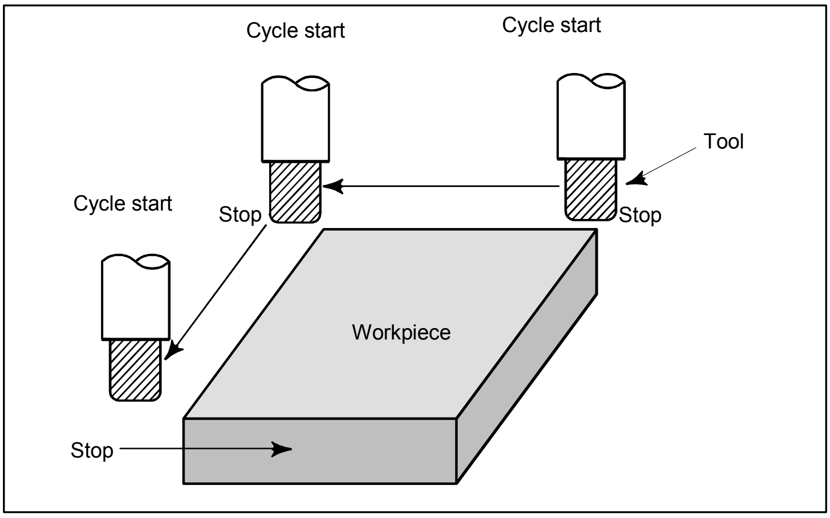


Fig. 5.5 (a) Single block

Single block

Procedure

1 Press the single block switch on the machine operator's panel.
The execution of the program is stopped after the current block is executed.

2 Press the cycle start button to execute the next block. The tool stops after the block is executed.
Refer to the appropriate manual provided by the machine tool builder for single block execution.

Explanation

- Reference position return and single block

If G28, G29, and G30 are issued, the single block function is effective at the intermediate point.

- Single block during a canned cycle

In a canned cycle, the single block stop points are the end of <1>, <2>, and <6> shown below. When the single block stop is made after the point <1> or <2>, the feed hold LED lights.

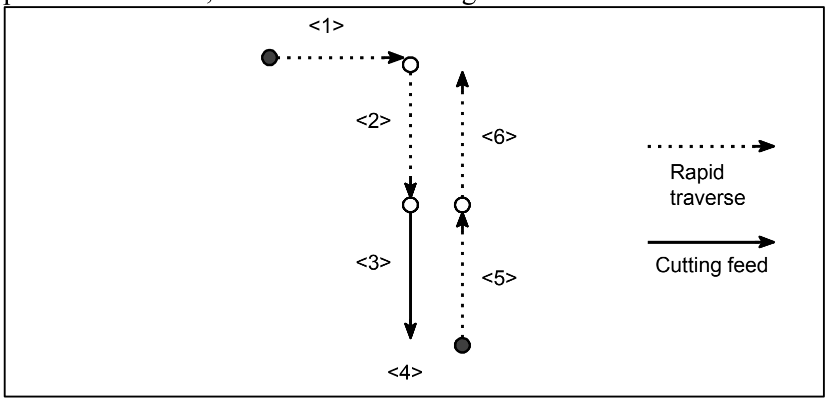


Fig. 5.5 (b) Single block during canned cycle

- Subprogram call and single block

Single block stop is not performed in a block containing M98P_;. M99; or G65.
However, single block stop is even performed in a block with M98P_ or M99 command, if the block contains an address other than O, N, P, L.

6 SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the tool from exceeding the stroke ends, Overtravel check and Stored stroke check are available. This chapter describes emergency stop, overtravel check, and stored stroke check.

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

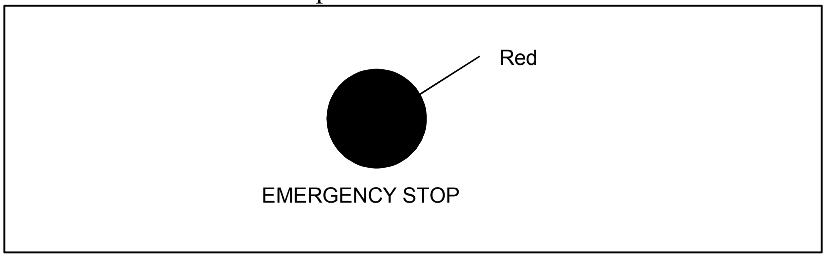


Fig. 6.1 (a) Emergency stop

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

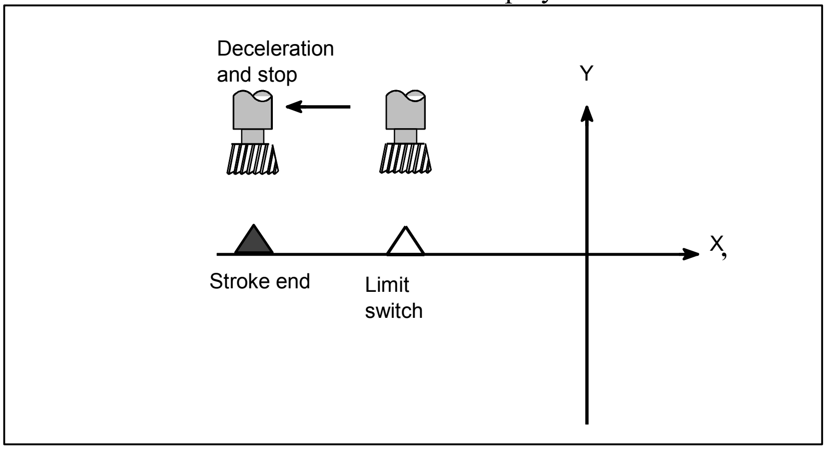


Fig. 6.2 (a) Overtravel

Explanation

- Overtravel during automatic operation

When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an overtravel alarm is displayed.

- Overtravel during manual operation

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

- Releasing overtravel

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

Alarm

Table6.2 (a)

Alarm No.	Message	Description
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.

6.3 STORED STROKE CHECK

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

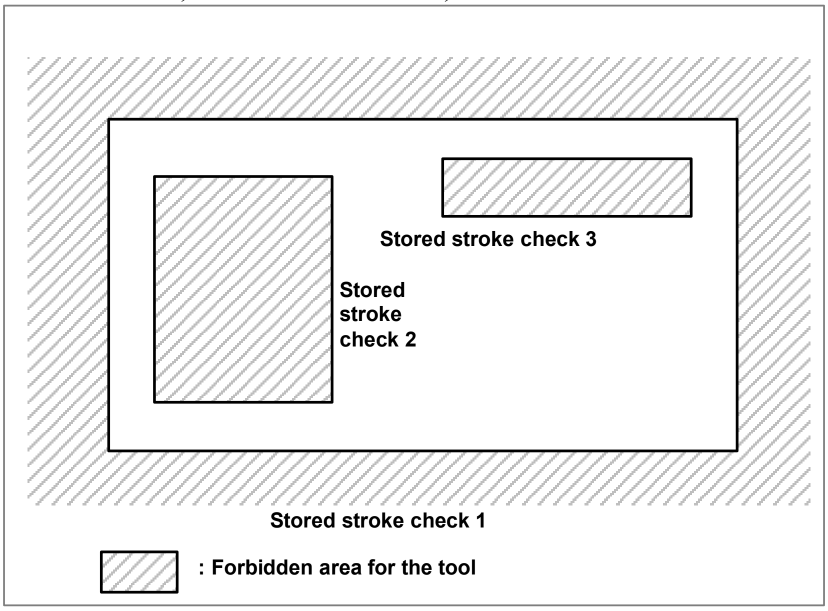


Fig. 6.3 (a) Stroke check

The following shows the areas which the tool cannot enter for each stored stroke check.

- Stored stroke check 1: Outside
- Stored stroke check 2: Outside or inside (switchable)
- Stored stroke check 3: Inside

When the tool moves into the forbidden area, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.
The stored stroke check 2 and 3 functions are optional.

Explanation

- Stored stroke check 1

Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.
At this time, a signal (overtravel alarm signal) can be output to the PMC if bit 6 (OTS) of parameter No. 1301 is set to 1. In addition, when the tool enters the forbidden area during manual operation, the signal (overtravel alarm signal) can be output to the PMC without generating the alarm by setting bit 1 (NAL) of parameter No. 1300 to 1. With this parameter setting, the alarm is generated when the tool enters the forbidden area during automatic operation.

> ⚠ **CAUTION**
> 1 If the two points for specifying a forbidden area are identical, all areas are handled as forbidden areas for stored stroke check 1.
> 2 The size of a forbidden area must be set carefully. If the size is set incorrectly, the stroke becomes infinite.

- Stored stroke check 2

Parameters (Nos. 1322, 1323) or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter OUT (No. 1300#0) selects either inside or outside as the forbidden area.
In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area.
Each of G22; and G23; should be commanded independently of another commands in a block.
The command below creates or changes the forbidden area:

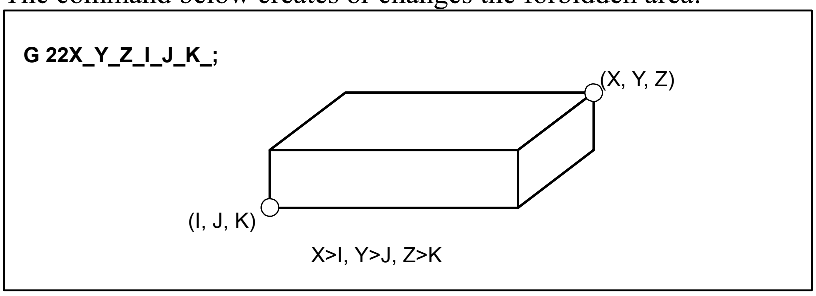


Fig. 6.3 (b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the figure below must be set.

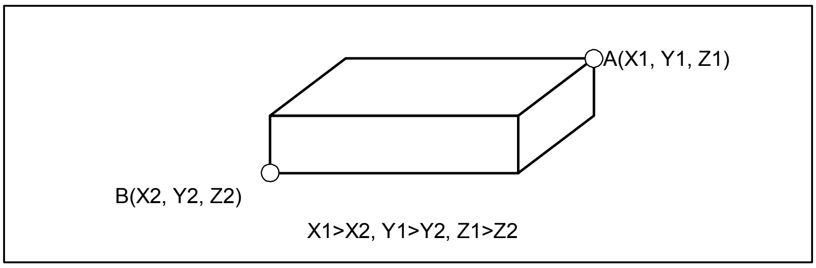


Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

The values X1, Y1, Z1, X2, Y2, and Z2, which are set by parameters No. 1322 and No. 1323, must be specified by the distance from the machine coordinate system (machine unit). The values X, Y, Z, I, J, and K, which are set by a G22 command, must be specified by the distance in the least input increment (input unit).
Values set by a program are then converted in the machine increment and the values are set as the parameters.

- Stored stroke check 3

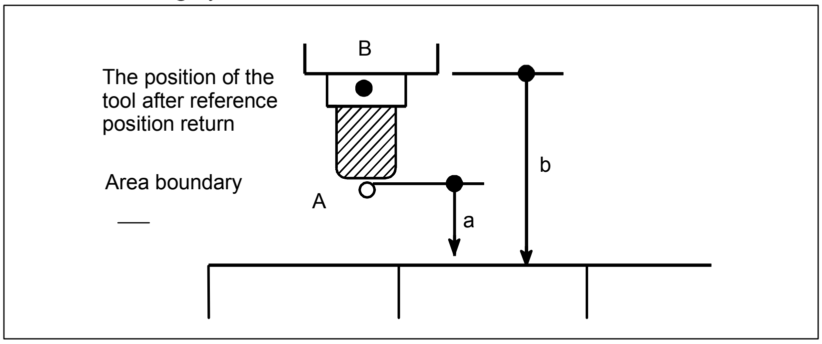
Set the boundary with parameters No. 1324 and 1325. The area inside the boundary becomes the forbidden area. The values X1, Y1, Z1, X2, Y2, and Z2 must be set as coordinates (machine unit) in the machine coordinate system.

> ⚠ **CAUTION**
> 1 If the two points for specifying a forbidden area are identical, all areas are handled as movable areas for stored stroke check 2/3.
> 2 Even if the two points for specifying a forbidden area are erroneously set, the rectangular parallelepiped having the points as vertices is assumed as a boundary.
> 3 Since an axis without the reference position return function has no forbidden areas, there are no alarms about forbidden areas for the axis.

- Checkpoints in the forbidden area

The parameter setting or programmed value (XYZIJK) depends on which part of the tool or tool holder is checked for entering the forbidden area.
If point A (the top of the tool) is checked in Fig. 6.3(d), the distance "a" should be set as the data for the stored stroke limit function. If point B (the tool chuck) is checked, the distance "b" must be set. When checking the tool tip (like point A), and if the tool length varies for each tool, setting the forbidden area for the longest tool requires no re-setting and results in safe operation.

- For milling system



- For lathe system

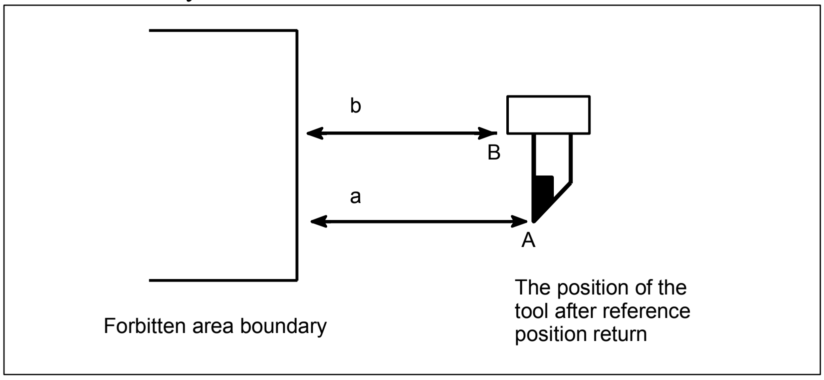


Fig. 6.3 (d) Setting the forbidden area

- Forbidden area overlapping

Area can be set in piles.

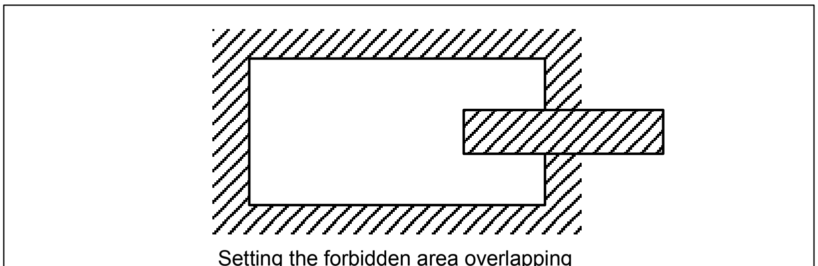


Fig. 6.3 (e) Setting the forbidden area overlapping

Unnecessary limits should be set beyond the machine stroke.

- Condition under which each check is enabled

Each check becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed.
After the power is turned on, if the reference position is in the forbidden area of each limit, an alarm is generated immediately. (Only in G22 mode for stored stroke check 2).

- Releasing the alarms

If the enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward.

- Change from G23 to G22 in a forbidden area

When G23 is switched to G22 in the forbidden area, the following results.

<1> When the forbidden area is inside, an alarm is informed in the next move.

<2> When the forbidden area is outside, an alarm is informed immediately.

- Timing for displaying an alarm

In stored stroke check 1/2/3, parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

Alarm

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	A movement in the positive direction exceeded stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	A movement in the negative direction exceeded stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	A movement in the positive direction exceeded stored stroke check 2.
OT0503	- OVERTRAVEL (SOFT 2)	A movement in the negative direction exceeded stored stroke check 2.
OT0504	+ OVERTRAVEL (SOFT 3)	A movement in the positive direction exceeded stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	A movement in the negative direction exceeded stored stroke check 3.

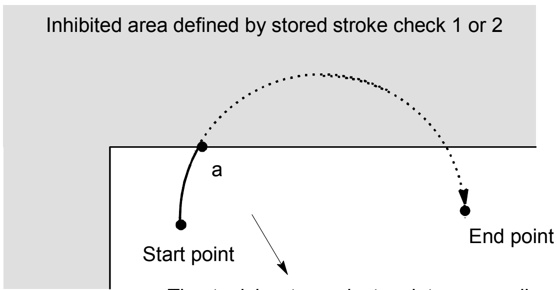
6.4 STROKE LIMIT CHECK BEFORE MOVE

During automatic operation, before the movement specified by a given block is started, whether the tool enters the inhibited area defined by stored stroke check 1, 2, or 3 is checked by determining the position of the end point from the current position of the machine and a specified amount of travel. If the tool is found to enter the inhibited area defined by a stored stroke limit, the tool is stopped immediately upon the start of movement for that block, and an alarm is displayed.

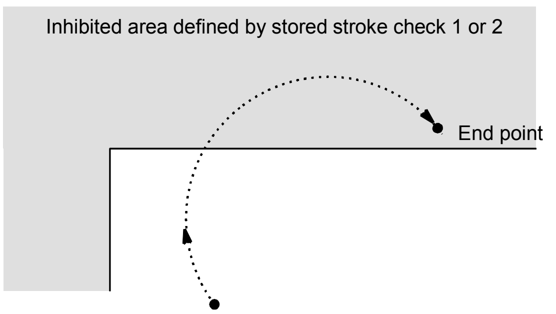
⚠ WARNING

Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a inhibited area is checked. In this case, the path followed by a move command is not checked. However, if the tool enters the inhibited area defined by stored stroke check 1, 2, or 3, an alarm is issued. (See the examples below.)

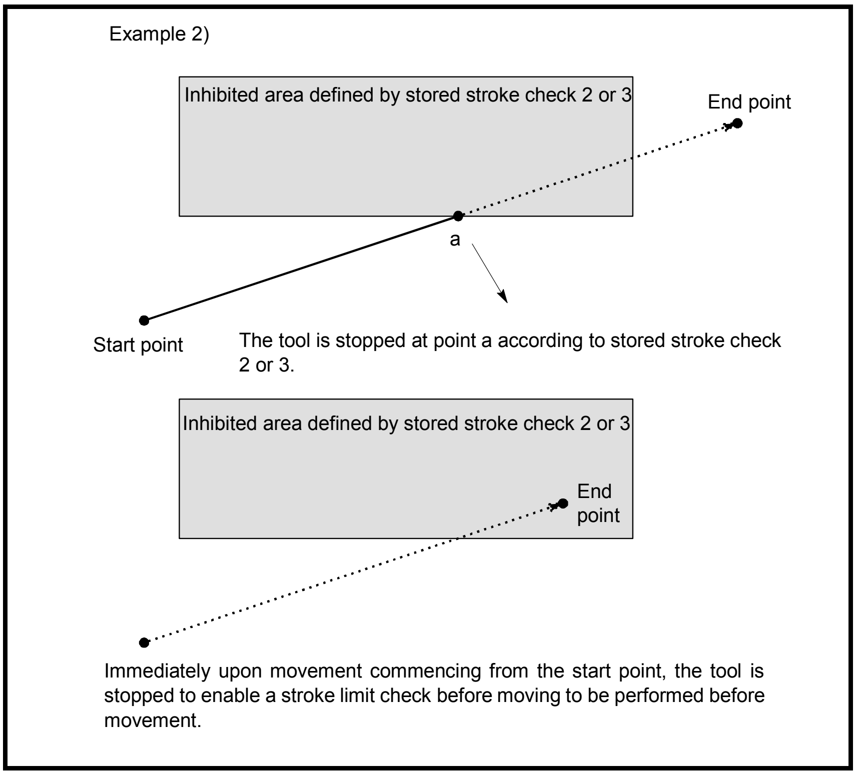
Example 1)



The tool is stopped at point a according to stored stroke check 1 or 2.



Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.



Explanation

When a stroke limit check before moving is performed, whether to check the movement performed by a G31 (skip) block and G37 (automatic tool length measurement) block can be determined using (parameter NPC (No. 1301#2)).

Limitation

- Machine lock

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- G23

When stored stroke check 2 is disabled (G23 mode), no check is made to determine whether the tool enters the inhibited area defined by stored stroke check 2.

- Program restart

When a program is restarted, an alarm is issued if the restart position is within a inhibited area.

- Manual intervention following a feed hold stop

When the execution of a block is restarted after manual intervention following a feed hold stop, no alarm is issued even if the end point following a manual intervention is within a inhibited area.

- A block consisting of multiple operations

If a block consisting of multiple operations (such as a canned cycle and exponential interpolation) is executed, an alarm is issued at the start point of any operation whose end point falls within a inhibited area.

- Cylindrical interpolation mode

In cylindrical interpolation mode, no check is made.

- Polar coordinate interpolation mode

In polar coordinate interpolation mode, no check is made.

- Three-dimensional coordinate conversion

In three-dimensional coordinate conversion mode, no check is made.

- PMC axis control

No check is made for a movement based on PMC axis control.

Alarm

Table 6.4 (a) Alarm

Number	Message	Description
OT0510	+ OVERTRAVEL (PRE-CHECK)	The block end point was found in the + side stroke limit prohibition area during a stroke check before movement. Modify the program.
OT0511	- OVERTRAVEL (PRE-CHECK)	The block end point was found in the - side stroke limit prohibition area during a stroke check before movement. Modify the program.

6.5 WRONG OPERATION PREVENTION FUNCTIONS

An improper tool offset setting or an improper operation of the machine can result in the workpiece being cut inadequately or the tool being damaged. Also, if data is lost due to an operation mistake, it takes extra time to recover from the mistake.

The operation confirmation functions described below are meant to prevent the operator from performing any unintended operation (hereinafter referred to as an improper operation).

1 Functions that are used when data is set
   - Data check to verify that the offset data is within the valid setting range
   - Incremental input operation confirmation
   - Prohibition of the absolute input by the soft key to prevent any improper absolute or incremental input operation
   - Confirmation of any operation of deleting the program or all data
   - Confirmation of a data update during the data setting process

2 Functions that are used when the program is executed
   - Highlighting of updated modal information
   - Display of the executed block status prior to the program execution
   - Display of the axis status, such as the mirror image function enabled or the interlock function enabled
   - Check for starting from the middle of the program
   - Data check to verify that the offset data is within the effective setting range
   - Maximum incremental value check

6.5.1 Functions that are Used When Data is Set

The following functions are provided to prevent improper operations when data is set.

- Input data range check
- Confirmation of incremental input
- Prohibition of the absolute input by the soft key
- Confirmation of the deletion of the program
- Confirmation of the deletion of all data
- Confirmation of a data update during the data setting process

Set these functions on the operation confirmation function setting screen. For the input data range check, set a valid input data range, e.g. the upper and lower limits, for each input screen. For the other functions, specify whether to enable or disable them.
For information about how to display the individual setting screens, how to manipulate them, and other details, see the item "Operation confirmation setting screen" that describes the operation procedures.

6.5.1.1 Input data range check

This function allows an effective data range to be set and checks whether the input data is within the set range.

Input data range check

Explanation

- Outline of the input data range check

This function allows an effective data range to be set for the data of each input screen listed later and checks whether the input data is within the set range. If the input data is out of the effective data range, the warning message "DATA IS OUT OF RANGE" is displayed and the data is rejected.

For example, assume that the effective data range for a certain tool offset number is set to -200. to 200, and that you are going to input 100.[INPUT]. Even if you inadvertently press the 0 key one more time, resulting in 1000.[INPUT], the input of 1000. is not accepted.
The function detects a setting mistake and prevents the program from running with invalid data.

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

- Settings

To enable this function, set an effective data range for each input screen on the operation confirmation function setting screen. For information about how to display the individual setting screens, how to set data ranges, and other details, see the items that describe the setting of the data ranges.
If the set data range is invalid, no data input is accepted. Correct the data range setting, and then input data.

- Disabling the function

The input data range check is disabled if you make any of the following settings on the operation confirmation function setting screen.

- Both the upper and lower limit values for the tool offset number or workpiece coordinate system are 0.
- The upper and lower limit values for each offset are identical.

- Messages displayed during the input data range check

When the cursor moves into an input field of an input screen, one of the following messages and warning messages is displayed. No message is displayed when the input data range check is disabled.

When the set effective data range is valid

Message list 1		
Input data status	**Message**	**Color**
The data in the input field is within the range.	Input range xxx - xxx	Black
The data in the input field is out of the range.	Input range xxx - xxx	Red

xxx: Upper and lower limit values

When the set effective data range is invalid

Message list 2		
Range check status	**Message**	**Color**
Tool offset number overlap	NG SETTING (OFFSET NUM OVERLAP)	Red
Workpiece coordinate system overlap	NG SETTING (WORK COORD VAL OVERLAP)	Red
Invalid upper and lower limit values	NG SETTING (U-LMT AND L-LMT ILLEGAL)	Red

The message "NG SETTING (U-LMT AND L-LMT ILLEGAL)" is displayed in the following cases:

- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

- Range check for data changed by G10 or system variable

If the data changed by G10 or system variable is out of the effective data range, the alarm PS0334 "OFFSET DATA OUT OF RANGE" is displayed.

6.5.1.2 Confirmation of incremental input

This function displays a confirmation message when you attempt to input an incremental value by using the [+INPUT] soft key.

Confirmation of incremental input

Explanation

- Outline of the confirmation of incremental input

This function displays a confirmation message when you attempt to input an incremental value by using the [+INPUT] soft key in any of the input screens listed below. It lets you confirm whether you really want to change data or not before making that change.

For example, when you set 5.[+INPUT] for 10., the message "15. INPUT OK?" is displayed.
The function prevents improper absolute or incremental input operations.

> **NOTE**
> This function cannot be used to input two or more values consecutively by delimiting them by commas (,).

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset
- Settings
- Parameter
- Pitch error compensation

M

- Chopping

T

- Workpiece shift
- Y-axis tool offset
- Second tool geometry offset
- Chuck tail stock barrier
- Tool geometry data

- Settings

In the operation confirmation function setting screen, check or uncheck the "INCREMENTAL INPUT" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.1.3 Prohibition of the absolute input by the soft key

This function prohibits the absolute input using the [INPUT] soft key.

Prohibition of the absolute input by the soft key

Explanation

- Outline of the prohibition of the absolute input by the soft key

This function prohibits the absolute input by the [INPUT] soft key in the input screens listed later.
It prevents improper absolute or incremental input operations by requiring that the absolute input be made using the INPUT MDI key and that the incremental input be made using the [+INPUT] soft key.

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

- Settings

In the operation confirmation function setting screen, check or uncheck the "DISABLED SOFTKEY[INPUT] IN" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.1.4 Confirmation of the deletion of the program

This function displays the confirmation message "DELETE PROGRAM ?" when you attempt to delete the program.

Confirmation of the deletion of the program

Explanation

- Outline of the confirmation of the deletion of the program

When you attempt to delete the program, this function displays the confirmation message "DELETE PROGRAM ?" It lets you confirm whether you really want to delete the program or not before executing the deletion.
The function prevents the program from being deleted due to an improper operation.

- Settings

In the operation confirmation function setting screen, check or uncheck the "PROGRAM DELETE" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.1.5 Confirmation of the deletion of all data

This function displays the confirmation message "DELETE ALL DATA?" when you attempt to delete all data.

Confirmation of the deletion of all data

Explanation

- Outline of the confirmation of the deletion of all data

When you attempt to delete all data on the input screen described later, this function displays the confirmation message "DELETE ALL DATA?". It lets you confirm whether you really want to delete all data or not before executing the deletion.
The function prevents all data from being deleted due to an improper operation.

- Input screens for which this function is effective

- Tool compensation

T

- Y-axis tool offset

- Settings

In the operation confirmation function setting screen, check or uncheck the "ALL DATA DELETE" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.1.6 Confirmation of a data update during the data setting process

This function displays the [CAN] and [EXEC] soft keys for confirmation when you attempt to update the data of an input screen during the data setting process.

Confirmation of a data update during the data setting process

Explanation

- Outline of the confirmation of a data update during the data setting process

When you input data in input screen during the data setting process, this function displays the [CAN] and [EXEC] soft keys for confirmation. It lets you confirm whether you really want to update the data or not before executing the update.
The function prevents set values from being lost due to an improper operation.

If you input data using the [+INPUT] soft key when the confirmation of incremental input is enabled, a message is displayed to confirm the incremental input.

- Settings

In the operation confirmation function setting screen, check or uncheck the "INPUT IN SETTING" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.2 Functions that are Used when the Program is Executed

Overview

The following functions are provided to prevent improper operations when the program is executed.

- Display of updated modal information
- Start check signal
- Axis status display
- Confirmation of the start from a middle block
- Data range check
- Maximum incremental value check

In the operation confirmation function setting screen, specify whether to enable or disable these functions individually.
For information about how to display the setting screen, how to manipulate it, and other details, see the item “Operation confirmation setting screen” that describes the operation procedures.

6.5.2.1 Display of updated modal information

This function allows modal information updated by the NC command or RESET to be highlighted in the modal information display for the current block.

Display of updated modal information

Explanation

- Outline of the display of updated modal information

This function allows modal information updated by the NC command or RESET to be highlighted in the modal information display for the current block.
For example, when a absolute command has been changed to an incremental command or when the workpiece coordinate system has been initialized by RESET, the function displays the changed part of the data in an easy-to-recognize manner, in order to prevent improper operations during the execution of the program.

- Settings

In the operation confirmation function setting screen, check or uncheck the "UPDATE MODAL HIGHLIGHT DISPLAY" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item "Operation confirmation setting" that describes the setting of the operation confirmation function.

6.5.2.2 Start check signal

This function displays the remaining amount of travel and modal information of the block to be executed and puts the program to a temporary halt before the program is executed.

Start check signal

Explanation

- Outline of the start check signal

When a cycle start is made with the start check signal STCHK <G0408 #0> set to 1, the function displays the remaining amount of travel and modal information of the block to be executed and puts the program to a temporary halt. Making the cycle start again resumes the execution of the program.
The function lets you check the status of the block before executing it, thus helping to prevent improper operations at the time of execution.
Using this function in combination with the updated modal information display function described in the preceding subsection makes it easier to check the status of the block to be executed.

- Settings

This function does not require any setting on the operation confirmation function setting screen.

6.5.2.3 Axis status display

This function displays the axis status to the left of the axis name in the coordinate display screen.

Axis status display

Explanation

- Outline of the axis status display

This function displays the axis status to the left of the axis name in the display of the machine coordinates, absolute coordinates, relative coordinates, and remaining travel amounts.
For example, when the mirror image function is enabled for the X1 axis, the absolute coordinates are displayed as follows.

```
ABSOLUTE
M  X1   10.000
   Y1   10.000
   Z1    0.000
```

By displaying the axis status as shown above, the function prevents improper operations at the time of execution.

- Axis status indication

The axis status is indicated as follows. These indications are listed in order of priority.

AXIS DETACH	: D
INTERLOCK	: I
MACHINE LOCK	: L
SERVO OFF	: S
AXIS MOVING	: *
MIRROR IMAGE	: M

- Settings

In the operation confirmation function setting screen, check or uncheck the "AXIS STATUS DISPLAY" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.2.4 Confirmation of the start from a middle block

This function displays a confirmation message when you attempt to execute a memory operation with the cursor placed on a block in the middle of the program.

Confirmation of the start from a middle block

Explanation

- Outline of the confirmation of the start from a middle block

This function displays the confirmation message "START FROM MIDDLE OF PROG (START/RESET)" when you attempt to execute a memory operation with the cursor placed on a block in the middle of the program. It lets you confirm whether you really want to start execution from that block or not before executing the program.
The function prevents you from inadvertently making a cycle start from a block in the middle of the program.

- Settings

In the operation confirmation function setting screen, check or uncheck the "START FROM MIDDLE OF PROGRAM" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.5.2.5 Data range check

This function lets you set an effective data range and check whether the data to be used for execution is within the set range.

Data range check

Explanation

- Outline of the data range check

This function lets you set an effective data range for each data item listed later and check whether the data to be used for execution is within the set range. If the data is out of the effective range, the alarm PS0334 "OFFSET DATA OUT OF EFFECTIVE RANGE" is displayed.
The function detects data setting mistakes and prevents the program from running with invalid data.

- Data for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

> **NOTE**
> To use this function, you need to set each effective data range correctly. For information about how to set the data ranges, see the item “Effective value range for each data”.

6.5.2.6 Maximum incremental value check

This function checks the maximum incremental value specified for each axis by the NC command.

Maximum incremental value check

Explanation

- Outline of the maximum incremental value check

When the maximum incremental value is specified by the NC command shown later, this function checks whether the amount of movement is kept below the specified value. If the specified value is exceeded, the alarm PS0337 "EXCESS MAXIMUM INCREMENTAL VALUE" is displayed.
A maximum incremental value can be specified on a per-axis basis and remains effective until 0 is set or the value is reset.
For example, when high precision contour control is used, the function checks whether the amount of movement between blocks is kept to the specified value or less. Through this process, it detects erroneous program settings and prevents the program from running with invalid data.

- Format

The format of the NC command used to specify the maximum incremental value is as follows.

G91.1 IP_ ;
IP_ ; Maximum incremental value

To cancel the maximum incremental value check, set 0.

6.5.3 Setting Screen

This section describes how to display the operation confirmation function setting screen and how to set the individual data items on this screen.
The operation confirmation function setting screen allows you to set the following items:

- Enabling or disabling each operation confirmation function
- Effective value range for the tool offset
- Effective value range for the workpiece origin offset

T

- Effective value range for the Y-axis tool offset
- Effective value range for the work shift

6.5.3.1 Operation confirmation function setting screen

This screen displays the enable/disable setting status of the following operation confirmation functions and lets you change their settings. (Hereinafter, the screen is referred to as the operation confirmation function setting screen.)

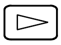
- Confirmation of incremental input
- Prohibition of the absolute input by the soft key
- Confirmation of the deletion of the program
- Confirmation of the deletion of all data
- Confirmation of a data update during the data setting process
- Display of updated modal information
- Axis status display
- Confirmation of the start from a middle block

Displaying and setting the operation confirmation function setting screen

Procedure

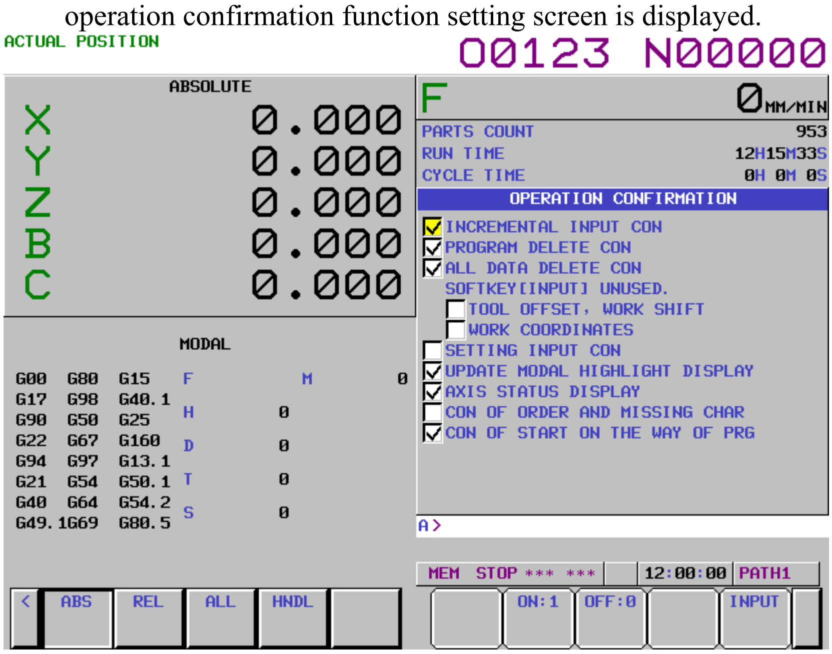
1 Press the 

function key.

2 Press the  soft key (continuous menu key) at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the operation confirmation function setting screen is displayed, click the [GUARD] soft key. The operation confirmation function setting screen is displayed.



Operation confirmation function setting screen

5 In the operation confirmation function setting screen, the check box of each enabled function is checked (V). Move the cursor to the check box of the item you want to set, by pressing the ↑, ↓, ←, and → keys.

6 Click the operation soft key [ON:1] or [OFF:0]. When you click the [ON:1] soft key, a check mark (V) appears in the corresponding check box, indicating that the function is enabled. When you click the [OFF:0] soft key, the check mark disappears from the check box, indicating that the function is disabled.

Explanation

- Items to be set

The following table shows what is displayed for each item to be set and the corresponding functions.

Displayed item	Corresponding function
INCREMENTAL INPUT	Confirmation of incremental input
DISABLED SOFTKEY[INPUT] IN TOOL OFFSET, WORK SHIFT	Prohibition of the absolute input by the soft key (tool offset, Y-axis tool offset (lathe system), and work shift (lathe system))
DISABLED SOFTKEY[INPUT] IN WORK COORDINATES	Prohibition of the absolute input by the soft key (workpiece origin offset)
PROGRAM DELETE	Confirmation of the deletion of the program
ALL DATA DELETE	Confirmation of the deletion of all data
INPUT IN SETTING	Confirmation of a data update during the data setting process
UPDATE MODAL HIGHLIGHT DISPLAY	Display of updated modal information
AXIS STATUS DISPLAY	Axis status display
START FROM MIDDLE OF PROGRAM	Confirmation of the start from a middle block

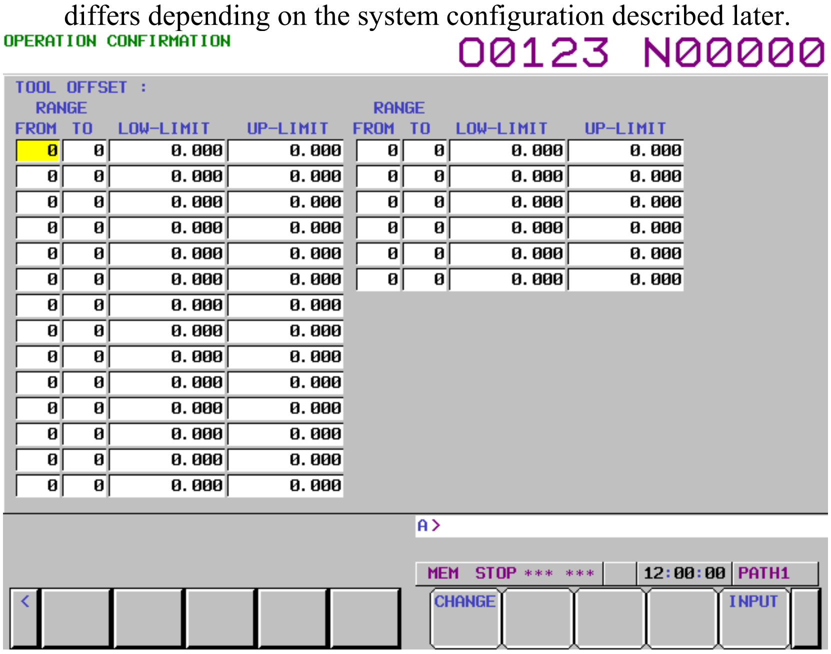
6.5.3.2 Tool offset range setting screen

This screen displays the setting status of tool offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the tool offset range setting screen.)
Up to 20 pairs of numbers can be specified to identify tool offset number ranges, and an effective offset value range can be defined for each of these 20 pairs.

Displaying and setting the tool offset range setting screen

Procedure

1 Press the OFFSET SETTING function key.
2 Press the ▷ soft key (continuous menu key) at the right edge of the screen several times until the [GUARD] soft key is displayed.
3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).
4 If any screen other than the tool offset range setting screen is displayed, click the [OFFSET] soft key. The tool offset range setting screen is displayed. What is displayed in this screen differs depending on the system configuration described later.



5 Move the cursor to the item you want to set, by using the PAGE ↑ and PAGE ↓ keys, ↑, ↓, ←, and → keys, or the [SWITCH] soft key.

6 Press the MDI key, enter necessary data, and then click the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a tool offset number overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the tool offset number are 0.
- The upper and lower offset limit values are identical.

Explanation

- System configuration

What to set differs for each of the following system configurations:

M

- Tool offset memory A
- Tool offset memory B
- Tool offset memory C

T

- Without geometry and wear offset
- With geometry and wear offset

M

- What to set with tool offset memory A

With tool offset memory A, an effective data range is specified using the following four items.

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
-	LOW-LIMIT	Specify a valid tool offset value range in connection with a specified tool offset number range.
	UP-LIMIT	

- What to set with tool offset memory B

With tool offset memory A, an effective data range is specified using the following six items.

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
GEOM	LOW-LIMIT	Specify a valid tool offset value range for geometry in connection with a specified tool offset number range.
	UP-LIMIT	
WEAR	LOW-LIMIT	Specify a valid tool offset value range for wear in connection with a specified tool offset number range.
	UP-LIMIT	

- What to set with tool offset memory C

With tool offset memory C, an effective data range is specified using the following ten items.

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	LENGTH	LOW-LIMIT	Specify a valid tool offset value range for geometry length in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for geometry radius in connection with a specified tool offset number range.
		UP-LIMIT	
WEAR	LENGTH	LOW-LIMIT	Specify a valid tool offset value range for wear length in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for wear radius in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this configuration, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

T

- What to set without geometry/wear offset

Without geometry/wear offset, an effective data range is specified using the following eight items.

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
X	LOW-LIMIT	Specify a valid tool offset value range for the X-axis in connection with a specified tool offset number range.
	UP-LIMIT	
Z	LOW-LIMIT	Specify a valid tool offset value range for the Z-axis in connection with a specified tool offset number range.
	UP-LIMIT	
RADIUS	LOW-LIMIT	Specify a valid tool offset value range for tool-nose radius in connection with a specified tool offset number range.
	UP-LIMIT	

NOTE
The radius-related items are not displayed if the cutter or tool nose radius compensation is not used.

- What to set with geometry/wear offset

With geometry/wear offset, an effective data range is specified using the following 12 items.

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	X	LOW-LIMIT	Specify a valid tool offset value range for the geometry X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z	LOW-LIMIT	Specify a valid tool offset value range for the geometry Z-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for geometry tool-nose radius in connection with a specified tool offset number range.
		UP-LIMIT	
WEAR	X	LOW-LIMIT	Specify a valid tool offset value range for the wear X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z	LOW-LIMIT	Specify a valid tool offset value range for the wear Z-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for wear tool-nose radius in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this system, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

NOTE
The radius-related items are not displayed if the tool nose radius compensation option is not displayed.

- Example of setting an input data range

For example, suppose that the following values are set with offset memory A.

FROM : TO	LOW-LIMIT : UP-LIMIT
1 : 20	0.000 : 100.000

In this case, the tool offset input screen accepts only offset values from 0.000 to 1000.000 for offset numbers 1 to 20.
If you attempt to input any other value, the warning message "DATA IS OUT OF RANGE" is displayed.

6.5.3.3 Workpiece origin offset range setting screen

This screen displays the setting status of workpiece origin offset and external workpiece origin offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the workpiece origin offset range setting screen.)

Up to six pairs of values can be specified to identify workpiece coordinate ranges for the workpiece origin offset, and an effective offset value range can be defined for each of the axes of these six pairs. As for the external workpiece origin offset, an effective offset value range can be specified for each axis.

Displaying and setting the workpiece origin offset range setting screen

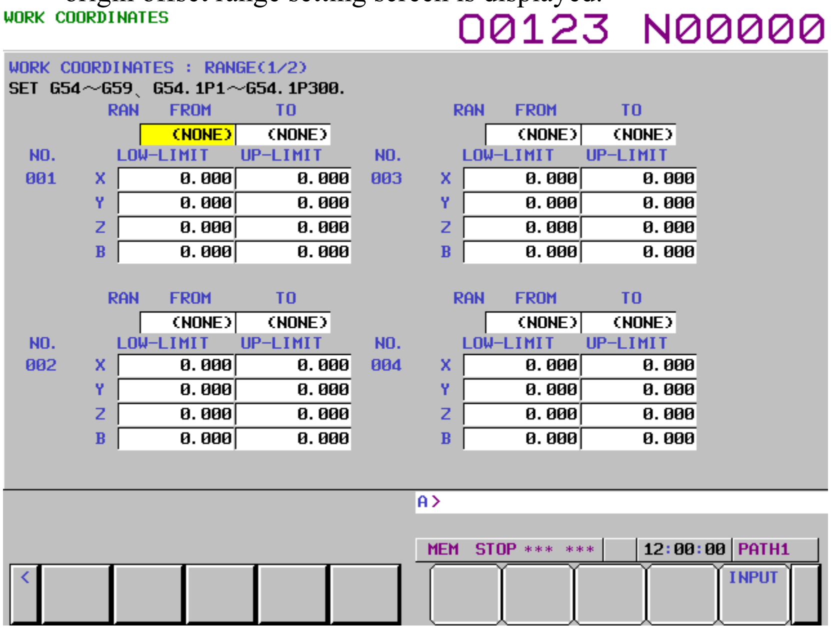
Procedure

1 Press the OFFSET SETTING function key.

2 Press the ▷ soft key (continuous menu key) at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the workpiece origin offset range setting screen is displayed, click the [WORK] soft key. The workpiece origin offset range setting screen is displayed.



Workpiece origin offset range setting screen

5 Move the cursor to the item you want to set, by using the [↑ PAGE] and [PAGE ↓] keys, [↑], [↓], [←], and [→] keys, or the [SWITCH] soft key.

6 Press the MDI key, enter necessary data, and then click the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a workpiece coordinate overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., an invalid workpiece coordinate system is set).
- The upper limit value is set for the workpiece coordinate system when 0 is set for the lower limit value.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the workpiece coordinate system are 0.
- The upper and lower limit values for each offset are identical.

Explanation

- What to set for the workpiece origin offset

For the workpiece origin offset, an effective data range is specified using the following four items.

Displayed item		What to set
RANGE	FROM	Specify a workpiece coordinate system range.
	TO	
AXIS NAME	LOW-LIMIT	Specify a valid offset value range in connection with a specified workpiece coordinate system range.
	UP-LIMIT	

- What to set for the external workpiece origin offset

For the external workpiece origin offset, an effective data range is specified using the following two items.

Displayed item		What to set
AXIS NAME	LOW-LIMIT	Specify a valid external workpiece origin offset value range on each axis.
	UP-LIMIT	

6.5.3.4 Y-axis tool offset range setting screen

T

In the case of a lathe system, this screen displays the setting status of Y-axis tool offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the Y-axis tool offset range setting screen.)
Up to four pairs of values can be specified to identify Y-axis tool offset number ranges, and an effective offset value range can be defined for each of these four pairs.

Displaying and setting the Y-axis tool offset range setting screen

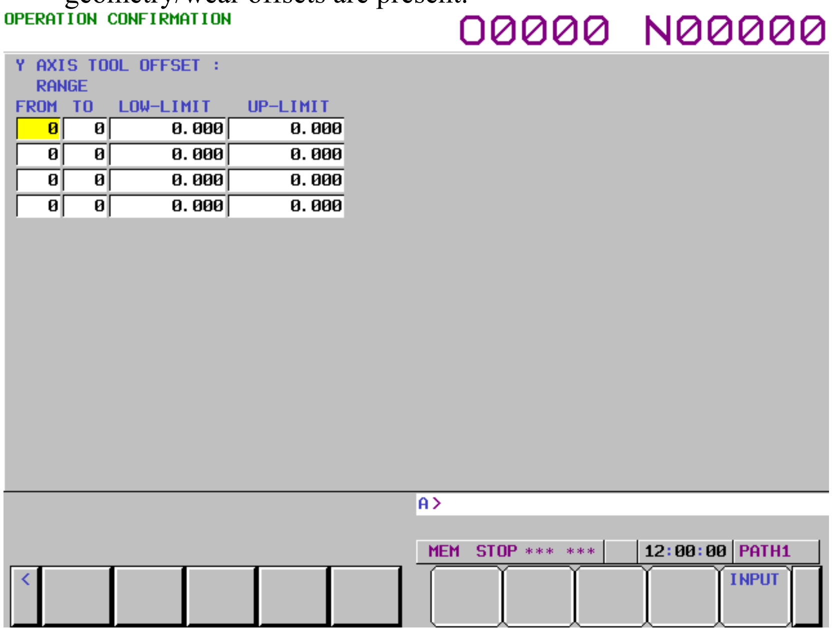
Procedure

1 Press the OFFSET SETTING function key.

2 Press the ▷ soft key (continuous menu key) at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the Y-axis tool offset range setting screen is displayed, click the [OFST.2] soft key. The Y-axis tool offset range setting screen is displayed. What is displayed in this screen differs depending on such factors as whether tool geometry/wear offsets are present.



Y-axis tool offset range setting screen

5 Move the cursor to the item you want to set, by using the ↑PAGE and PAGE↓ keys, ↑, ↓, ←, and → keys, or the [SWITCH] soft key.

6 Press the MDI key, enter necessary data, and then click the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a tool offset number overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the tool offset number are 0.
- The upper and lower offset limit values are identical.

Explanation

- What to set without geometry/wear offsets

Without geometry/wear offsets, an effective data range is specified using the following four items.

Displayed item		What to set
RANGE	FROM	Specify a Y-axis tool offset number range.
	TO	
-	LOW-LIMIT	Specify a valid tool offset value range in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	

- What to set with geometry/wear offsets

With geometry/wear offsets, an effective data range is specified using the following six items.

Displayed item		What to set
RANGE	FROM	Specify a Y-axis tool offset number range.
	TO	
GEOM	LOW-LIMIT	Specify a valid tool offset value range for geometry in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	
WEAR	LOW-LIMIT	Specify a valid tool offset value range for wear in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	

6.5.3.5 Workpiece shift range setting screen

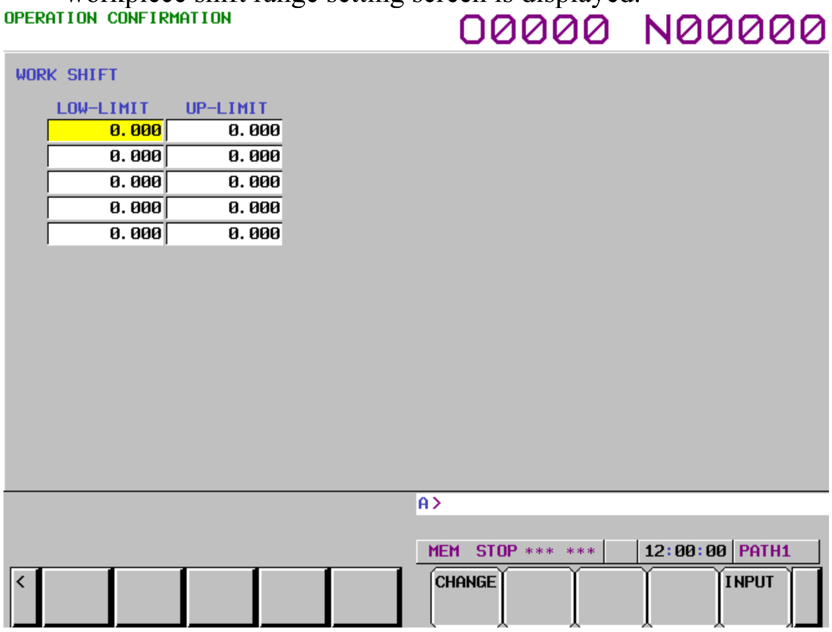
T

In the case of a lathe system, this screen displays the setting status of shift effective data ranges of workpiece shift coordinate systems and lets you change their settings. (Hereinafter, the screen is referred to as the workpiece shift range setting screen.)
An offset value range can be specified for each axis.

Displaying and setting workpiece shift input ranges

Procedure

1 Press the OFFSET SETTING function key.
2 Press the ▷ soft key (continuous menu key) at the right edge of the screen several times until the [GUARD] soft key is displayed.
3 Click the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).
4 If any screen other than the workpiece shift range setting screen is displayed, click the [WORK SHIFT] soft key. The workpiece shift range setting screen is displayed.



Workpiece shift range setting screen

5 Move the cursor to the item you want to set, by using the ↑ PAGE and PAGE ↓ keys, ↑, ↓, ←, and → keys, or the [SWITCH] soft key.
6 Press the MDI key, enter necessary data, and then click the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- The upper and lower limit values are reversed.

Also, the input data range check is invalidated in the following cases.

- The upper and lower offset limit values are identical.

Explanation

- What to set for the workpiece shift

For the workpiece shift, an effective data range is specified using the following two items.

Displayed item		What to set
AXIS NAME	LOW-LIMIT	Specify a valid workpiece shift coordinate system shift value range on each axis.
	UP-LIMIT	

7 ALARM AND SELF-DIAGNOSIS FUNCTIONS

When an alarm occurs, the corresponding alarm screen appears to indicate the cause of the alarm. The causes of alarms are classified by error codes and number. Up to 60 previous alarms can be stored and displayed on the screen (alarm history display).
The system may sometimes seem to be at a halt, although no alarm is displayed. In this case, the system may be performing some processing. The state of the system can be checked using the self-diagnosis function.

7.1 ALARM DISPLAY

Explanation

- Alarm screen

When an alarm is issued, the display changes to the alarm screen. Two alarm screens "DETAIL" and "ALL PATH" are provided. You can choose one of the screens by pressing the corresponding soft key.

- Detail screen
  Alarm information for the currently selected path is displayed.

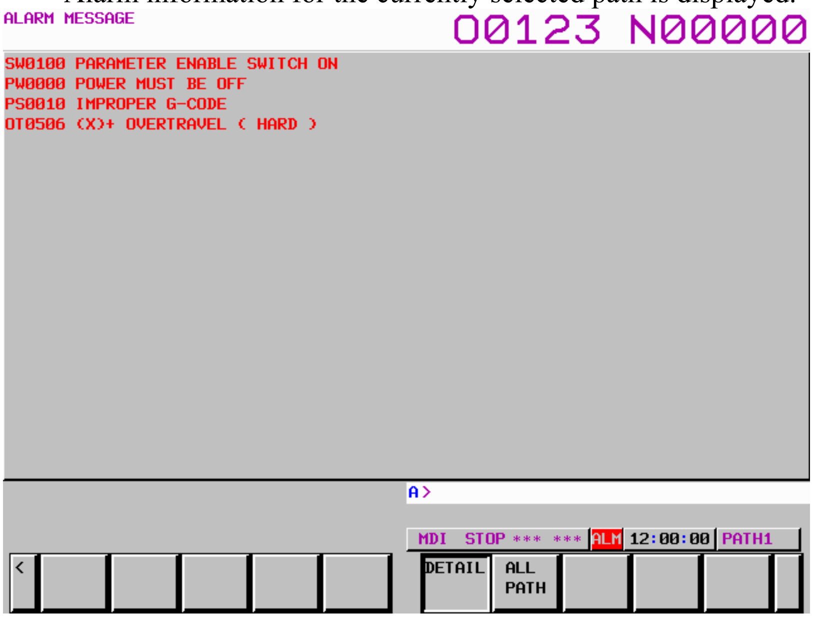


Fig. 7.1 (a) Alarm detail screen

- All path screen
  Alarm information for all paths is displayed sequentially from path 1.

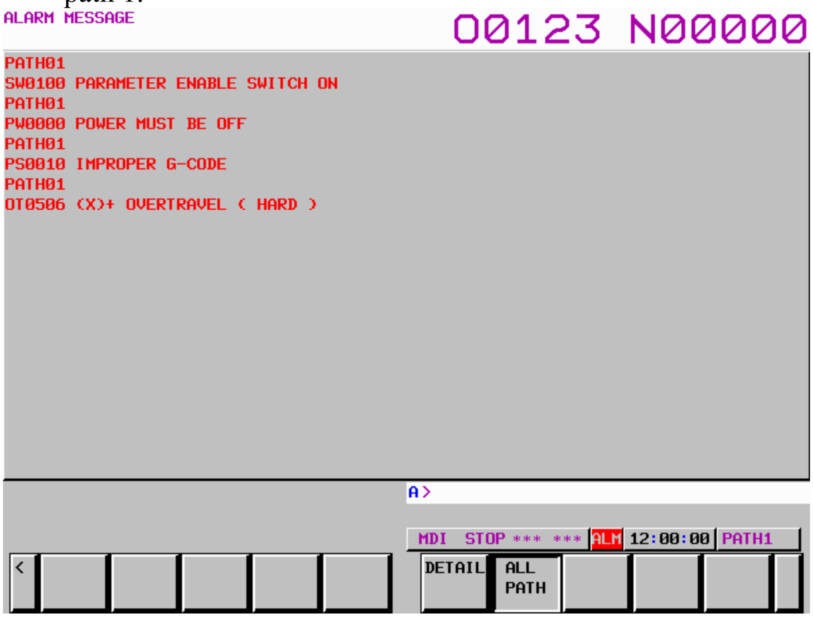


Fig. 7.1 (b) All path screen

- Displaying an alarm screen

ALM is sometimes indicated in the bottom part of the screen display without displaying an alarm screen.

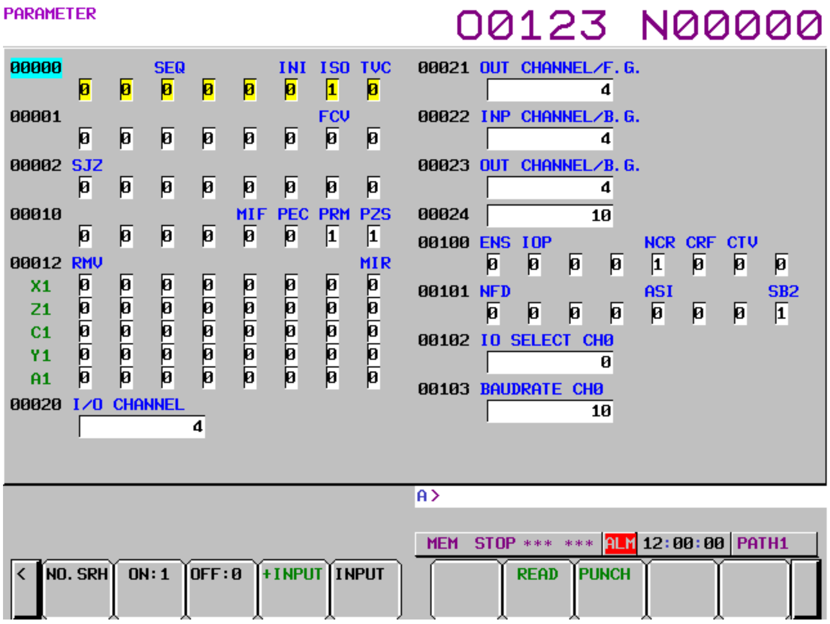


Fig. 7.1 (c) Parameter screen

In this case, display the alarm screen by following the steps below.

1 Press the 

function key.

2 Press the [ALARM] chapter selection soft key.

3 Pressing the [ALARM] soft key changes the screen display to the "DETAIL" screen (or the alarm screen selected previously), and the [DETAIL] and [ALL PATH] soft keys appears.

   Pressing the [DETAIL] soft key displays the "DETAIL" screen.

   Pressing the [ALL PATH] soft key displays the "ALL PATH" screen.

   If the number of paths is 1, pressing the [ALARM] soft key displays the "DETAIL" screen, but the [ALARM] soft key indication remains unchanged.

4 You can change pages by using the page key.

- Releasing alarm

The cause of an alarm can be determined from the error code, number, and associated message. To release the alarm, generally correct the cause, then press the reset key.

- Error code and number

The type of an alarm is indicated by an error code and number.
Example: PS0010, SV0004, etc.
For details, see Appendix G, "ALARMS".

7.2 ALARM HISTORY DISPLAY

Up to 60 alarms (in 10 screen pages) issued by the CNC including the latest alarm are stored and displayed on the screen.
The display procedure is explained below.

Alarm history display

Procedure

1 Press the MESSAGE function key.

2 Press the [HISTRY] chapter selection soft key.
An alarm history is displayed.
The following information is displayed:
<1> Date and time of alarm issuance
<2> Alarm type
<3> Alarm number
<4> Alarm message (sometimes not displayed depending on the alarm)
<5> Page No.

3 You can change pages by using the page key.

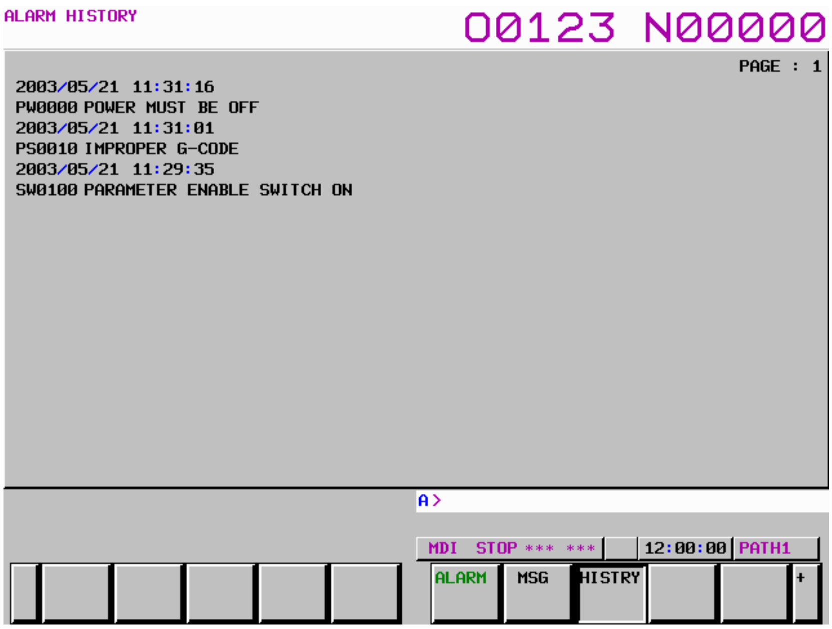


Fig. 7.2 (a) Alarm history screen

7.3 CHECKING BY SELF-DIAGNOSIS SCREEN

The system may sometimes seem to be at a halt, although no alarm has occurred. In this case, the system may be performing some processing. The state of the system can be checked by displaying the self-diagnosis screen.

Procedure for Diagnois

Procedure

1 Press the SYSTEM function key.
2 Press the [DGNOS] chapter selection soft key.
3 The diagnosis screen has more than 1 pages. Select the screen by the following operation.
   (1) Change the page by the page change key.
   (2) Method by soft key
      - Key input the number of the diagnosis data to be displayed.
      - Press [N SRCH].

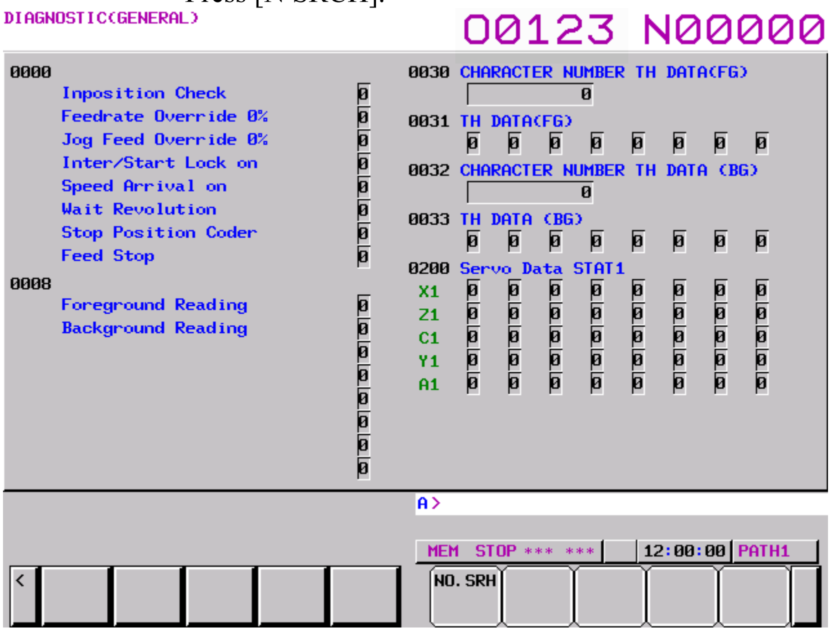


Fig. 7.3 (a) Self-diagnosis screen

8 DATA INPUT/OUTPUT

By using the memory card interface on the left side of the display, information written in a memory card is read into the CNC and information is written from the CNC to a memory card.
The following types of data can be input and output:

1. Program
2. Offset data
3. Parameter
4. Pitch error compensation data
5. Three-dimensional error compensation data
6. Custom macro common variable
7. Workpiece coordinate system setting data
8. Operation history data
9. Tool management data

The above data can be input and output on the screens used for displaying and setting the data and on the ALL IO screen.

8.1 INPUT/OUTPUT ON EACH SCREEN

Various types of data including programs, parameters, offsets, pitch error compensation data, macro variables, workpiece coordinate system data, operation history data, and tool management data can be input and output using operation screens.

8.1.1 Inputting and Outputting a Program

8.1.1.1 Inputting a program

The following explains how to input a program from a memory card to the memory of the CNC by using the program editing screen or program directory screen.

Inputting a program

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator’s panel.
3 Press function key PROG , then the program editing screen or program directory screen appears.
4 Press soft key [(OPRT)].
5 Press the rightmost soft key ▻ (continuous menu key).
6 Press soft key [READ].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
8 Press soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

8.1.1.2 Outputting a program

A program stored in the memory of the CNC unit is output to a memory card.

Outputting a program

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [PROG], then the program editing screen or program directory screen appears.
4 Press soft key [(OPRT)].
5 Press the rightmost soft key [▷] (continuous menu key).
6 Press soft key [PUNCH].
7 Type the number of the program that you want to output.
  If you type "O-9999" or nothing, all programs in the memory are output to "ALL-PROG.TXT".
8 Press soft key [EXEC].
  This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
  To cancel the output, press soft key [CANCEL].

8.1.2 Inputting and Outputting Parameters

8.1.2.1 Inputting parameters

Parameters are loaded into the memory of the CNC unit from a memory card. The input format is the same as the output format. When a parameter is loaded which has the same data number as a parameter already registered in the memory, the loaded parameter replaces the existing parameter.

Inputting parameters

Procedure

1 Make sure the input device is ready for reading.
2 Press function key OFFSET SETTING .
3 Press the soft key [SETTING] for chapter selection, then the setting screen appears.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
Alarm SW0100 appears.
5 Press function key SYSTEM .
6 Press chapter selection soft key [PARAM], then the parameter screen appears.
7 Press the EDIT switch on the machine operator’s panel.
8 Press soft key [(OPRT)].
9 Press soft key [READ].
10 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
11 Press soft key [EXEC].
This starts reading the program, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press soft key [CANCEL].
12 Press function key OFFSET SETTING .
13 Press the chapter selection soft key [SETTING].
14 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
15 Turn the power to the CNC back on.

8.1.2.2 Outputting parameters

All parameters are output in the defined format from the memory of the CNC to a memory card.

Outputting parameters

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press function key SYSTEM .
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press chapter selection soft key [PARAM], then the parameter screen appears.
6 Press soft key [(OPRT)].
7 Press soft key [PUNCH].
8 Type the file name that you want to output.
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
9 Press soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

Explanation

- Suppressing output of parameters set to 0

When bit 1 (PRM) of parameter No. 0010 is set to 1, and [EXEC] is pressed, the following parameters are not output:

	Other than axis type	Axis type
Bit type	Parameter for which all bits are set to 0.	Parameter for an axis for which all bits are set to 0.
Value type	Parameter whose value is 0.	Parameter for an axis for which the value is 0.

8.1.3 Inputting and Outputting Offset Data

8.1.3.1 Inputting offset data

Offset data is loaded into the memory of the CNC from a memory card. The input format is the same as for offset value output. When an offset value is loaded which has the same offset number as an offset number already registered in the memory, the loaded offset data replaces existing data.

Inputting offset data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING], then the tool compensation screen appears.
4 Press soft key [(OPRT)].
5 Press the rightmost soft key [▷] (continuous menu key).
6 Press soft key [READ].
7 Type the name of the file that you want to input.
   If the input file name is omitted, default input file name "TOOLOFST.TXT" is assumed.
8 Press soft key [EXEC].
   This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
   To cancel the input of the program, press soft key [CANCEL].

8.1.3.2 Outputting offset data

All offset data is output in a output format from the memory of the CNC to a memory card.

Outputting offset data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key OFFSET SETTING , then the tool compensation screen appears.
4 Press soft key [(OPRT)].
5 Press the rightmost soft key ▷ (continuous menu key).
6 Press soft key [PUNCH].
7 Type the file name that you want to output.
If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the offset data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

Explanation
- Output format

M

Output format is as follows:

- Tool compensation memory A

```
%
G10 G90 P01 R_ Q_
G10 G90 P02 R_ Q_
 ...
G10 G90 P_ R_
%
```

Q_ : Virtual tool nose number (TIP). Not output when the virtual tool nose direction is not used.
P_ : Tool offset number (1 to the number of tool compensation pairs)
R_ : Tool compensation data. Output with a decimal point in the input unit used at output.

- Tool compensation memory B

```
%
G10 G90 L10 P01 R_ Q_
G10 G90 L11 P01 R_
G10 G90 L10 P02 R_ Q_
 ...
G10 G90 L11 P_ R_
%
```

L10 : Geometric compensation amount
L11 : Wear compensation amount
Q_, P_, and R_ have the same meanings as for tool compensation memory A.

- Tool compensation memory C

```
%
G10 G90 L10 P01 R_ Q_
G10 G90 L11 P01 R_
G10 G90 L12 P01 R_
G10 G90 L13 P01 R_
G10 G90 L10 P02 R_ Q_
 ...
G10 G90 L12 P_ R_
G10 G90 L13 P_ R_
%
```

L10 : Geometry compensation amount corresponding to the H code

L11 : Wear compensation amount corresponding to the H code

L12 : Geometry compensation amount corresponding to the D code

L13 : Wear compensation amount corresponding to the D code

Q_, P_, and R_ have the same meanings as for tool compensation memory A.

T

The tool compensation amount and tool nose radius compensation amount are output in the following format.

%
G10 P01 X_ Z_ R_ Q_ Y_
G10 P02 X_ Z_ R_ Q_ Y_
...
G10 P__ X_ Z_ R_ Q_ Y_
G10 P10001 X_ Z_ R_ Y_
G10 P10002 X_ Z_ R_ Y_
...
G10 P100__ X_ Z_ R_ Y_
%

P_: Tool compensation number (1 to the number of tool compensation pairs)
Tool offset number: Specification of the tool compensation amount and tool wear compensation amount
10000 + tool offset number: Specification of the tool geometry compensation amount

X_: Tool compensation data (X). Output with a decimal point in the input unit used at output.

Z_: Tool compensation data (Z). Same as X_.

R_: Tool nose radius offset amount (R). The data format is the same as for X_.
When tool nose radius compensation is not provided, this item is not output.

Q_: Virtual tool nose number (TIP). When tool nose radius compensation is not provided, this item is not output.

Y_ : Tool compensation data (Y). The data format is the same as for X_.
When no Y-axis offset is provided, this item is not output.

The second tool geometry compensation amount is output in the following format.

```
%
G10 P20001 X_ Z_ Y_
G10 P20002 X_ Z_ Y_

G10 P200__ X_ Z_ Y_
%
```

P_ : Tool compensation number (1 to the number of tool compensation pairs)
Tool offset number: Specification of the tool compensation amount and tool wear compensation amount
20000 + tool offset number: Specification of the second tool geometry compensation amount

The other addresses are the same as for the tool compensation amount.

NOTE
The input format and output format do not depend on the G-code system A/B/C.

8.1.4 Inputting and Outputting Pitch Error Compensation Data

8.1.4.1 Inputting pitch error compensation data

Pitch error compensation data are loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When a pitch error compensation data is loaded which has the corresponding data number as a pitch error compensation data already registered in the memory, the loaded data replaces the existing data.

Inputting pitch error compensation data

Procedure

1 Make sure the input device is ready for reading.
2 Press function key OFFSET SETTING .
   Press the chapter selection soft key [SETTING].
3 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
   Alarm SW0100 appears.
4 Press function key SYSTEM .
5 Press the rightmost soft key ▷ (continuous menu key) and press chapter selection soft key [PITCH].
6 Press the EDIT switch on the machine operator's panel.
7 Press soft key [(OPRT)].
8 Press the rightmost soft key ▷ (continuous menu key).
9 Press soft key [READ].
10 Type the name of the file that you want to input.
   If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
11 Press soft key [EXEC].
   This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
   To cancel the input of the program, press soft key [CANCEL].
12 Press function key OFFSET SETTING .
13 Press the chapter selection soft key [SETTING].
14 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
15 Turn the power to the CNC back on.

8.1.4.2 Outputting pitch error compensation data

All pitch error compensation data are output in the defined format from the memory of the CNC to a memory card.

Outputting pitch error compensation data

Procedure

1. Make sure the output device is ready for output.
2. Press function key SYSTEM .
3. Press the rightmost soft key ▷ (continuous menu key) and press chapter selection soft key [PITCH].
4. Press soft key [(OPRT)].
5. Press the rightmost soft key ▷ (continuous menu key).
6. Press the EDIT switch on the machine operator's panel.
7. Press soft key [PUNCH].
8. Type the file name that you want to output.
   If the file name is omitted, default file name "PITCH.TXT" is assumed.
9. Press soft key [EXEC].
   This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
   To cancel the output, press soft key [CANCEL].

8.1.4.3 Input/output format of pitch error compensation data

Pitch error compensation data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Keyword	Meaning of the following numeric value
N	Pitch error compensation data number + 10000
Q	Data identification (1 : Parameter data, 0 : Pitch error compensation data)
P	Pitch error compensation data value

- Format

Pitch error compensation data is output in the following format:

N	*****	Q0	P	****	;

The 5-digit numeric value following N indicates a pitch error compensation data number to which a value of 10000 is added.
Q0 indicates pitch error compensation data
The numeric value following P indicates the value (integer value) of pitch error compensation data between -128 to 127.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

> **Example**
> N10001Q0P100;
> Pitch error compensation data number 1
> Pitch error compensation data value 100

- Beginning and end of a record

A pitch error compensation data record begins with % and ends with %.

> **Example**
> % ; .... Beginning of record
> N10000Q0P10;
> N10001Q0P100;
> :
> N11279Q0P0;
> % .... End of record

When parameters and pitch error compensation data are integrated into one file, % is added to the beginning and end of the file.

8.1.5 Inputting and Outputting Three-dimensional Error Compensation Data

8.1.5.1 Inputting three-dimensional error compensation data

Three-dimensional error compensation data are loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When a three-dimensional error compensation data is loaded which has the corresponding data number as a three-dimensional error compensation data already registered in the memory, the loaded data replaces the existing data.

Inputting three-dimensional error compensation data

Procedure

1 Make sure the input device is ready for reading.

2 Press function key OFFSET SETTING .
Press the chapter selection soft key [SETTING].

3 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
Alarm SW0100 appears.

4 Press function key SYSTEM .

5 Press the rightmost soft key ▷ (continuous menu key) and press chapter selection soft key [3D ERR COMP].

6 Press the EDIT switch on the machine operator’s panel.

7 Press soft key [(OPRT)].

8 Press the rightmost soft key ▷ (continuous menu key).

9 Press soft key [READ].

10 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "COMP3D.TXT" is assumed.

11 Press soft key [EXEC].
This starts reading the three-dimensional error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

12 Press function key OFFSET SETTING .

13 Press the chapter selection soft key [SETTING].

14 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.

15 Turn the power to the CNC back on.

8.1.5.2 Outputting three-dimensional error compensation data

All three-dimensional error compensation data are output in the defined format from the memory of the CNC to a memory card.

Outputting three-dimensional error compensation data

Procedure

1 Make sure the output device is ready for output.
2 Press function key SYSTEM .
3 Press the rightmost soft key ▷ (continuous menu key) and press chapter selection soft key [3D ERR COMP].
4 Press soft key [(OPRT)].
5 Press the rightmost soft key ▷ (continuous menu key).
6 Press the EDIT switch on the machine operator's panel.
7 Press soft key [PUNCH].
8 Type the file name that you want to output.
If the file name is omitted, default file name "COMP3D.TXT" is assumed.
9 Press soft key [EXEC].
This starts outputting the three-dimensional error compensation data, and "OUTPUT" blinks in the lower right part of the screen.
When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.5.3 Input/output format of three-dimensional error compensation data

Three-dimensional error compensation data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Keyword	Meaning of the following numeric value
N	Data number (compensation point number + 100000) as represented with six digits
A1	First compensation axis
A2	Second compensation axis
A3	Third compensation axis
P	Compensation data (-128 to 127)

- Format

Three-dimensional error compensation data is output in the following format:

N	*****	A1	P	****	A2	P	****	A3	P	****	;

The 6-digit numeric value following N indicates a three-dimensional error compensation data number to which a value of 100000 is added.
The numeric value following P indicates the value (integer value) of three-dimensional error compensation data between -128 to 127.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example

N100001A1P100A2P110A3P120;

Three-dimensional error compensation data number 1

Error compensation data value for the 1st compensation axis 100

Error compensation data value for the 2nd compensation axis 110

Error compensation data value for the 3rd compensation axis 120

- Beginning and end of a record

A three-dimensional error compensation data record begins with % and ends with %.

Example

```
% ;        .....................................Beginning of record
N100001 A1 P1 A2 P2 A3 P3 ;
N100002 A1 P0 A2 P0 A3 P-3 ;
   :
N115625 A1 P1 A2 P1 A3 P0 ;
%          ............................................. End of record
```

- Input of compensation data using G10

Compensation data can be changed from a machining program, using the programmable parameter input function.

The command format is as follows:

%
G10 L51 ;
N_ P_ R_ ;
N_ P_ R_ ;
:
G11 ;
%

G10 L51	:	Three-dimensional error compensation data input mode
G11	:	Cancellation of three-dimensional error compensation data input mode
N	:	Compensation point number (1-15625)
P	:	Compensation axis number (1-3)
R	:	Compensation data (-128-127)

NOTE

1 To input compensation data using G10, the option of the programmable parameter input function is required.
2 In three-dimensional error compensation data input mode, no other NC statements can be issued.
3 The decimal point cannot be used in address N, P, and R.

8.1.6 Inputting and Outputting Custom Macro Common Variables

8.1.6.1 Inputting custom macro common variables

The value of a custom macro common variable is loaded into the memory of the CNC from a memory card. The same format used to output custom macro common variables is used for input.

Inputting custom macro common variables

Procedure

1. Make sure the input device is ready for reading.
2. Press function key OFFSET SETTING .
3. Press the rightmost soft key ▷ (continuous menu key) and press chapter selection soft key [MACRO].
4. Press soft key [(OPRT)].
5. Press the rightmost soft key ▷ (continuous menu key).
6. Press the EDIT switch on the machine operator's panel.
7. Press soft key [READ].
8. Type the name of the file that you want to input.
   If the input file name is omitted, default input file name "MACRO.TXT" is assumed.
9. Press soft key [EXEC].
   This starts reading the custom macro common variables, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
   To cancel the input of the program, press soft key [CANCEL].

Explanation

- Common variables

The common variables (#500 to #549) can be input and output.
(When the option for adding a common variable is specified, values from #500 to #999 can be input and output.) #100 to #149 can be input when bit 3 (PV5) of parameter No. 6001 is set to 1. (When the option for adding a common variable is specified, values from #100 to #199 can be input and output.)

8.1.6.2 Outputting custom macro common variables

Custom macro common variables stored in the memory of the CNC can be output in the defined format to a memory card.

Outputting custom macro common variables

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key OFFSET SETTING .
4 Press the rightmost soft key ▷ (continuous menu key) and press chapter selection soft key [MACRO].
5 Press soft key [(OPRT)].
6 Press the rightmost soft key ▷ (continuous menu key).
7 Press soft key [PUNCH].
8 Type the file name that you want to output.
If the file name is omitted, default file name "MACRO.TXT" is assumed.
9 Press soft key [EXEC].
This starts outputting the custom macro common variables, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

Explanation

- Output format

The output format is as follows:
The values of custom macro variables are output in a bit-image hexadecimal representation of double-precision floating-point type data.

```
%
G10L85P200(0000000000000000)
G10L85P200(0000000000000000)
G10L85P200(FFFFFFFFFFFFFFFF)
  :
G10L85P500(4024000000000000)
G10L85P501(4021000000000000)
G10L85P502(0000000000000000)
  :
SETVN500[ABC,DEF]
SETVN501[GHI,JKL]
SETVN502[MNO,PQR]
  :
M02
%
```

NOTE
The conventional custom macro statement program format cannot be used for output.

- Common variable

The common variables (#500 to #549) can be input and output.
(When the option for adding a common variable is specified, values from #500 to #999 can be input and output.) #100 to #149 can be output when bit 3 (PV5) of parameter No. 6001 is set to 1. (When the option for adding a common variable is specified, values from #100 to #199 can be input and output.)

8.1.7 Inputting and Outputting Workpiece Coordinates System Data

8.1.7.1 Inputting workpiece coordinate system data

Coordinate system variable data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When coordinate system variable data with a data number corresponding to existing coordinate system variable data registered in the memory is loaded, the loaded coordinate system variable data replaces the existing coordinate system variable data.

Inputting workpiece coordinate system data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the coordinate system variable screen.
4 Press soft key [(OPRT)].
5 Press the rightmost soft key [▷] (continuous menu key).
6 Press soft key [READ].
7 Type the name of the file that you want to input.
   If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
8 Press soft key [EXEC].
   This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
   To cancel the input of the program, press soft key [CANCEL].

8.1.7.2 Outputting workpiece coordinate system data

All coordinate system variable data is output in the output format from the memory of the CNC to a memory card.

Outputting workpiece coordinate system data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key OFFSET SETTING to display the coordinate system variable screen.
4 Press soft key [(OPRT)].
5 Press the rightmost soft key ▷ (continuous menu key).
6 Press soft key [PUNCH].
7 Type the file name that you want to output.
If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.8 Inputting and Outputting Operation History Data

Only output operation is permitted on operation history data.
The output data is in text format. So, to reference the output data you must use an application that can handle text files on the personal computer.

8.1.8.1 Outputting operation history data

All operation history data is output in the output format form the memory of the CNC to a memory card.

Outputting operation history data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator’s panel.
3 Press function key [SYSTEM] to display the operation history screen.
1 Press soft key [(OPRT)].
2 Press the rightmost soft key [▷] (continuous menu key).
3 Press soft key [PUNCH].
4 Type the file name that you want to output.
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
5 Press soft key [EXEC].
This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.9 Inputting and Outputting Tool Management Data

> **NOTE**
> 1 For multi-path systems, place all paths in the EDIT mode before performing input and output operations.
> 2 The format used is the same as the registration format of the G10 format.

8.1.9.1 Inputting tool management data

Tool management data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When tool management data with a data number corresponding to existing tool management data registered in the memory is loaded, the loaded tool management data replaces the existing tool management data.

Inputting tool management data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the tool management screen or magazine screen.
4 Press soft key [(OPRT)].
5 Press soft key [READ].
6 Press soft key [TOOL].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOL_MNG.TXTT" is assumed.
8 Press soft key [EXEC].
This starts reading the tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

> **NOTE**
> When using large diameter tool support of the tool management function, keep the following in mind.
> - If a target tool is registered in a cartridge and interferes with other tools in registration or modification of tool figure data of the tool management data, PS alarm 5360 is issued. (The data is not input.)
> - When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool figure data, tool management data, and cartridge management table data in this order.

8.1.9.2 Outputting tool management data

All tool management data is output in the output format from the memory of the CNC to a memory card.

Outputting tool management data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key OFFSET SETTING to display the tool management screen or magazine screen.
4 Press soft key [(OPRT)].
6 Press soft key [PUNCH].
5 Press soft key [TOOL].
7 Type the file name that you want to output.
If the file name is omitted, default file name "TOOL_MNG.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the tool management data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.9.3 Inputting magazine data

Magazine data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When magazine data with a data number corresponding to existing magazine data registered in the memory is loaded, the loaded magazine data replaces the existing magazine data.

Inputting magazine data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the tool management screen or magazine screen.
4 Press soft key [(OPRT)].
5 Press soft key [READ].
6 Press soft key [MAGAZINE].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "MAGAZINE.TXT" is assumed.
8 Press soft key [EXEC].
This starts reading the magazine data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

NOTE

When using large diameter tool support of the tool management function, keep the following in mind.。

- If a tool interferes with other tools in registration in or modification to the cartridge management table, PS alarm 5360 is issued. (The data is not input.)
- When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool figure data, tool management data, and cartridge management table data in this order.

8.1.9.4 Outputting magazine data

All magazine data is output in the output format from the memory of the CNC to a memory card.

Outputting magazine data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key OFFSET SETTING to display the tool management screen or magazine screen.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Press soft key [MAGAZINE].
7 Type the file name that you want to output.
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.9.5 Inputting tool life status name data

Tool life status name data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When tool life status name data with a data number corresponding to existing tool life status name data registered in the memory is loaded, the loaded tool life status name data replaces the existing tool life status name data.

Inputting tool life status name data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the tool management screen or magazine screen.
4 Press soft key [(OPRT)].
5 Press soft key [READ].
6 Press soft key [STATUS].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "STATUS.TXT" is assumed.
8 Press soft key [EXEC].
This starts reading the tool life status name data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

8.1.9.6 Outputting tool life status name data

All tool life status name data is output in the output format from the memory of the CNC to a memory card.

Outputting tool life status name data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator’s panel.
3 Press function key [OFFSET SETTING] to display the tool management screen or magazine screen.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Press soft key [STATUS].
7 Type the file name that you want to output.
If the file name is omitted, default file name "STATUS.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the tool life status name data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.9.7 Inputting name data of customize data

Name data of customize data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When name data of customize data with a data number corresponding to existing name data of customize data registered in the memory is loaded, the loaded name data of customize data replaces the existing data.

Inputting name data of customize data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator’s panel.
3 Press function key [OFFSET SETTING] to display the tool management screen or magazine screen.
4 Press soft key [(OPRT)].
5 Press soft key [READ].
6 Press soft key [CUSTOM].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CUSTOMIZ.TXT" is assumed.
8 Press soft key [EXEC].
This starts reading the name data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

8.1.9.8 Outputting name data of customize data

All name data of customize data is output in the output format from the memory of the CNC to a memory card.

Outputting name data of customize data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator’s panel.
3 Press function key [OFFSET SETTING] to display the tool management screen or magazine screen.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Press soft key [CUSTOM].
7 Type the file name that you want to output.
If the file name is omitted, default file name "CUSTOMIZ.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the name data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.9.9 Inputting customize data displayed as tool management data

Customize data displayed as tool management data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When customize data displayed as tool management data with a data number corresponding to existing customize data displayed as tool management data registered in the memory is loaded, the loaded customize data replaces the existing data.

Inputting customize data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key OFFSET SETTING to display the tool management screen, magazine screen, or each tool data screen.
4 Press soft key [(OPRT)].
5 Press soft key [READ].
6 Press soft key [CUSTOMIZE SCREEN].
7 Type the name of the file that you want to input.
   If the input file name is omitted, default input file name "DISPCSTM.TXT" is assumed.
8 Press soft key [EXEC].
   This starts reading the customize data displayed as tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
   To cancel the input of the program, press soft key [CANCEL].

8.1.9.10 Outputting customize data displayed as tool management data

Customize data displayed as tool management data is output from the memory of the CNC to a memory card in the output format.

Outputting customize data displayed as tool management data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the tool management screen, magazine screen, or each tool data screen.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Press soft key [CUSTOMIZE SCREEN].
7 Type the file name that you want to output.
  If the file name is omitted, default file name "DISPCSTM.TXT" is assumed.
8 Press soft key [EXEC].
  This starts outputting the customize data displayed as tool management data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
  To cancel the output, press soft key [CANCEL].

8.1.9.11 Inputting spindle waiting position name data

Spindle waiting position name data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When spindle waiting position name data with a data number corresponding to existing spindle waiting position name data registered in the memory is loaded, the loaded spindle waiting position name data replaces the existing data.

Inputting spindle waiting position name data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key OFFSET SETTING to display the tool management screen, magazine screen, or each tool data screen.
4 Press soft key [(OPRT)].
5 Press soft key [READ].
6 Press soft key [SPINDLE WAITING NAME].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "POSNAME.TXT" is assumed.
8 Press soft key [EXEC].
This starts reading the spindle waiting position name data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

8.1.9.12 Outputting spindle waiting position name data

Spindle waiting position name data is output from the memory of the CNC to a memory card in the output format.

Outputting spindle waiting position name data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key OFFSET SETTING to display the tool management screen, magazine screen, or each tool data screen.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Press soft key [SPINDLE WAITING NAME].
7 Type the file name that you want to output.
If the file name is omitted, default file name "POSNAME.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the spindle waiting position name data display, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.9.13 Inputting decimal point position data of customize data

Decimal point position data of customize data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When decimal point position data of customize data with a data number corresponding to existing decimal point position data of customize data registered in the memory is loaded, the loaded decimal point position data of customize data replaces the existing data.

Inputting decimal point position data of customize data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the tool management screen, magazine screen, or each tool data screen.
4 Press soft key [(OPRT)].
5 Press soft key [READ].
6 Press soft key [DECIMAL POINT DATA].
7 Type the name of the file that you want to input.
  If the input file name is omitted, default input file name "POINTPOS.TXT" is assumed.
8 Press soft key [EXEC].
  This starts reading the decimal point position data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
  To cancel the input of the program, press soft key [CANCEL].

8.1.9.14 Outputting decimal point position data of customize data

Decimal point position data of customize data is output from the memory of the CNC to a memory card in the output format.

Outputting decimal point position data of customize data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the tool management screen, magazine screen, or each tool data screen.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Press soft key [DECIMAL POINT DATA].
7 Type the file name that you want to output.
If the file name is omitted, default file name "POINTPOS.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the decimal point position data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.1.9.15 Inputting tool geometry data

Tool geometry data is loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When tool geometry data with a data number corresponding to existing tool geometry data registered in the memory is loaded, the loaded tool geometry data replaces the existing data.

Inputting tool geometry data

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the tool management screen, magazine screen, each tool data screen, or tool geometry data screen.
4 Press soft key [(OPRT)].
5 Press soft key [READ].
6 Press soft key [TOOL GEOMETRY DATA].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLGEOM.TXT" is assumed.
8 Press soft key [EXEC].
This starts reading the tool geometry data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

NOTE
1 If the tool with a number of tool geometry data to be changed is registered to the magazine when an attempt is made to change the tool geometry data, an alarm (PS5360) is issued. (The data is not input.)
2 After data related to the tool management functions in the NC has all been cleared, restore backup data in the following order: Tool geometry data, tool management data, and magazine management table.

8.1.9.16 Outputting tool geometry data

Tool geometry data is output from the memory of the CNC to a memory card in the output format.

Outputting tool geometry data

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press function key [OFFSET SETTING] to display the tool management screen, magazine screen, each tool data screen, or tool geometry data screen.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Press soft key [TOOL GEOMETRY DATA].
7 Type the file name that you want to output.
If the file name is omitted, default file name "TOOLGEOM.TXT" is assumed.
8 Press soft key [EXEC].
This starts outputting the tool geometry data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.2 INPUT/OUTPUT ON THE ALL IO SCREEN

Just by using the ALL IO screen, you can input and output programs, parameters, offset data, pitch error compensation data, macro variables, workpiece coordinate system data, operation history data, and tool management data.

The following explains how to display the ALL IO screen:

Displaying the ALL IO screen

Procedure

1 Press function key SYSTEM .
2 Press the rightmost soft key ▷ (continuous menu key) several times.
3 Press soft key [ALL IO] to display the ALL IO screen.

The subsequent steps to select data from the ALL IO screen will be explained for each type of data.

8.2.1 Inputting/Outputting a Program

A program can be input and output using the ALL IO screen.

Inputting a program

Procedure

1 Press soft key [PRGRM] on the ALL IO screen.
2 Select EDIT mode.
3 Press soft key [(OPRT)].
4 Press soft key [N READ].
5 Set the name of the file that you want to input.
  Type a file name, and press soft key [F NAME].
  If the input file name is omitted, default file name "ALL-PROG.TXT" is assumed.
6 Set the program number to be used after the input.
  Type a program number, and press soft key [0 SET].
  If the program number is omitted, the program number in the file is used directly.
7 Press soft key [EXEC].
  This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
  To cancel the input of the program, press soft key [CANCEL].

Outputting a program

Procedure

1 Press soft key [PRGRM] on the ALL IO screen.
2 Select EDIT mode.
3 Press soft key [(OPRT)].
4 Press soft key [PUNCH].
5 Set the program that you want to output.
  Type a program number, and press soft key [0 SET].
  If -9999 is typed, all programs in the memory are output.
6 Set the file name to be output.
  Type a file name, and press soft key [F NAME].
  When no file name is set, the output file name is assumed to be "O-number.TXT" if a single program number is specified; if -9999 is specified, the output file name is assumed to be "ALL-PROG.TXT".
7 Press soft key [EXEC].
  This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
  To cancel the output, press soft key [CANCEL].

8.2.2 Inputting and Outputting Parameters

Parameters can be input and output using the ALL IO screen.

Inputting parameters

Procedure

1 Press function key [OFFSET SETTING] .
2 Press soft key [SETTING].
3 Enter 1 in response to the prompt for “PARAMETER WRITE” in setting data. Alarm SW0100 appears.
4 Press soft key [PARAM] on the ALL IO screen.
5 Select EDIT mode.
6 Press soft key [(OPRT)].
7 Press soft key [N READ].
8 Set the name of the file that you want to input.
Type a file name, and press soft key [F NAME].
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
9 Press soft key [EXEC].
This starts reading the parameter, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].
10 Press function key [OFFSET SETTING] .
11 Press the chapter selection soft key [SETTING].
12 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
13 Turn the power to the CNC back on.

Outputting parameters

Procedure

1 Press soft key [PARAM] on the ALL IO screen.
2 Select EDIT mode.
3 Press soft key [(OPRT)].
4 Press soft key [PUNCH].
5 Set the file name to be output.
Type a file name, and press soft key [F NAME].
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
6 Press soft key [EXEC].
This starts outputting the program, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press soft key [CANCEL].

8.2.3 Inputting and Outputting Offset Data

Offset data can be input and output using the ALL IO screen.

Inputting offset data

Procedure

1 Press soft key [OFFSET] on the ALL IO screen.
2 Select EDIT mode.
3 Press soft key [(OPRT)].
4 Press soft key [N READ].
5 Set the name of the file that you want to input.
   Type a file name, and press soft key [F NAME].
   If the input file name is omitted, default input file name "TOOLOFST.TXT" is assumed.
6 Press soft key [EXEC].
   This starts reading the offset data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
   To cancel the input of the program, press soft key [CANCEL].

Outputting offset data

Procedure

1 Press soft key [OFFSET] on the ALL IO screen.
2 Select EDIT mode.
3 Press soft key [(OPRT)].
4 Press soft key [PUNCH].
5 Set the file name to be output.
   Type a file name, and press soft key [F NAME].
   If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
6 Press soft key [EXEC].
   This starts outputting the offset data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
   To cancel the output, press soft key [CANCEL].

8.2.4 Inputting/Outputting Pitch Error Compensation Data

Pitch error compensation data can be input and output using the ALL IO screen.

Inputting pitch error compensation data

Procedure

1 Press function key [OFFSET SETTING].
2 Press soft key [SETTING].
3 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100 appears.
4 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
5 Press soft key [PITCH].
6 Select EDIT mode.
7 Press soft key [(OPRT)].
8 Press soft key [N READ].
9 Set the name of the file that you want to input.
Type a file name, and press soft key [F NAME].
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
10 Press soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].
11 Press function key [OFFSET SETTING].
12 Press the chapter selection soft key [SETTING].
13 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
14 Turn the power to the CNC back on.

Outputting pitch error compensation data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [PITCH].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Set the file name to be output.
Type a file name, and press soft key [F NAME].
If the file name is omitted, default file name "PITCH.TXT" is assumed.
7 Press soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.2.5 Inputting/Outputting Custom Macro Common Variables

Custom macro common variables can be input and output using the ALL IO screen.

Inputting custom macro common variables

Procedure

1 Press soft key [MACRO] on the ALL IO screen.
2 Select EDIT mode.
3 Press soft key [(OPRT)].
4 Press soft key [N READ].
5 Set the name of the file that you want to input.
Type a file name, and press soft key [F NAME].
If the input file name is omitted, default input file name "MACRO.TXT" is assumed.
6 Press soft key [EXEC].
This starts reading the custom macro common variables, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

Outputting custom macro common variables

Procedure

1 Press soft key [MACRO] on the ALL IO screen.
2 Select EDIT mode.
3 Press soft key [(OPRT)].
4 Press soft key [PUNCH].
5 Set the file name to be output.
Type a file name, and press soft key [F NAME].
If the file name is omitted, default file name "MACRO.TXT" is assumed.
6 Press soft key [EXEC].
This starts outputting the custom macro common variables, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.2.6 Inputting and Outputting Workpiece Coordinates System Data

Workpiece coordinates system data can be input and output using the ALL IO screen.

Inputting workpiece coordinate system data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [WORK].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [N READ].
6 Set the name of the file that you want to input.
  Type a file name, and press soft key [F NAME].
  If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
7 Press soft key [EXEC].
  This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
  To cancel the input of the program, press soft key [CANCEL].

Outputting workpiece coordinate system data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [WORK].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Set the file name to be output.
  Type a file name, and press soft key [F NAME].
  If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
7 Press soft key [EXEC].
  This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
  To cancel the output, press soft key [CANCEL].

8.2.7 Inputting and Outputting Operation History Data

Operation history data can be output using the ALL IO screen.
Only output operation is permitted for operation history data.
The output data is in text format. So, to reference the output data you must use an application that can handle text files on the personal computer.

Outputting operation history data

Procedure

1 Press the rightmost soft key ▷ (continuous menu key) several times on the ALL IO screen.
2 Press soft key [OPEHIS].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Set the file name to be output.
Type a file name, and press soft key [F NAME].
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
7 Press soft key [EXEC].
This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.2.8 Inputting and Outputting Tool Management Data

Tool management data can be input and output using the ALL IO screen.

> **NOTE**
> 1 For multi-path systems, place all paths in the EDIT mode before performing input and output operations.
> 2 The format used is the same as the registration format of the G10 format.

Inputting tool management data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [TOOL].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [N READ].
6 Set the name of the file that you want to input.
Type a file name, and press soft key [F NAME].
If the input file name is omitted, default input file name "TOOL_MNG.TXT" is assumed.
7 Press soft key [EXEC].
This starts reading the tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

Outputting tool management data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [TOOL].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Set the file name to be output.
Type a file name, and press soft key [F NAME].
If the file name is omitted, default file name "TOOL_MNG.TXT" is assumed.
7 Press soft key [EXEC].
This starts outputting the tool management data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

Inputting magazine data

Procedure

1. Press the rightmost soft key ▷ (continuous menu key) several times on the ALL IO screen.
2. Press soft key [MAGAZINE].
3. Select EDIT mode.
4. Press soft key [(OPRT)].
5. Press soft key [N READ].
6. Set the name of the file that you want to input.
   Type a file name, and press soft key [F NAME].
   If the input file name is omitted, default input file name "MAGAZINE.TXT" is assumed.
7. Press soft key [EXEC].
   This starts reading the magazine data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
   To cancel the input of the program, press soft key [CANCEL].

Outputting magazine data

Procedure

1. Press the rightmost soft key ▷ (continuous menu key) several times on the ALL IO screen.
2. Press soft key [MAGAZINE].
3. Select EDIT mode.
4. Press soft key [(OPRT)].
5. Press soft key [PUNCH].
6. Set the file name to be output.
   Type a file name, and press soft key [F NAME].
   If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
7. Press soft key [EXEC].
   This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
   To cancel the output, press soft key [CANCEL].

Inputting tool life status name data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [STATUS].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [N READ].
6 Set the name of the file that you want to input.
Type a file name, and press soft key [F NAME].
If the input file name is omitted, default input file name "STATUS.TXT" is assumed.
7 Press soft key [EXEC].
This starts reading the tool life status name data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

Outputting tool life status name data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [STATUS].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Set the file name to be output.
Type a file name, and press soft key [F NAME].
If the file name is omitted, default file name "STATUS.TXT" is assumed.
7 Press soft key [EXEC].
This starts outputting the tool life status name data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

Inputting name data of customize data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [CUSTOM].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [N READ].
6 Set the name of the file that you want to input.
Type a file name, and press soft key [F NAME].
If the input file name is omitted, default input file name "CUSTOMIZ.TXT" is assumed.
7 Press soft key [EXEC].
This starts reading the name data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press soft key [CANCEL].

Outputting name data of customize data

Procedure

1 Press the rightmost soft key [▷] (continuous menu key) several times on the ALL IO screen.
2 Press soft key [CUSTOM].
3 Select EDIT mode.
4 Press soft key [(OPRT)].
5 Press soft key [PUNCH].
6 Set the file name to be output.
Type a file name, and press soft key [F NAME].
If the file name is omitted, default file name "CUSTOMIZ.TXT" is assumed.
7 Press soft key [EXEC].
This starts outputting the name data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press soft key [CANCEL].

8.2.9 File Format and Error Messages

Explanation

- File format

All files that are read from and written to a memory card are of text format. The format is described below.

A file starts with % or LF, followed by the actual data. A file always ends with %. In a read operation, data between the first % and the next LF is skipped. Each block ends with an LF, not a semicolon (;).

- LF: 0A (hexadecimal) of ASCII code
- When a file containing lowercase letters, kana characters, and several special characters (such as $, \, and !) is read, those letters and characters are ignored.

Example)

```
%
O0001(MEMORY CARD SAMPLE FILE)
G17 G49 G97
G92 X-11.3 Y2.33
  :
  :
M30
%
```

- ASCII code is used for input/output, regardless of the setting parameter (ISO/EIA).
- Bit 3 of parameter No. 0100 can be used to specify whether the end of block code (EOB) is output as "LF" only, or as "LF, CR, CR."

Limitation

- Memory card specification

Use memory cards that comply with PCMCIA Ver2.0 or JEIDA Ver4.1.

- Attribute memory

Memory cards that have no attribute memory or contain no device information in the attribute memory cannot be used.

- Flash ROM card

Flash ROM cards can be used for loading only.

8.3 EMBEDDED ETHERNET OPERATIONS

8.3.1 FTP File Transfer Function

The operation of the FTP file transfer function is described below.

Host file list display

A list of the files held on the host computer is displayed.

Procedure

1 Press the function key PROG .
2 Press the [FOLDER] soft key. The program list screen appears. (If the soft key does not appear, press the continuous menu key.)
3 Press the [(OPRT)] soft key.
4 Press the [DEVICE CHANGE] soft key. And the displayed device will be changed. Until the host file list screen appears, press the key a few times.

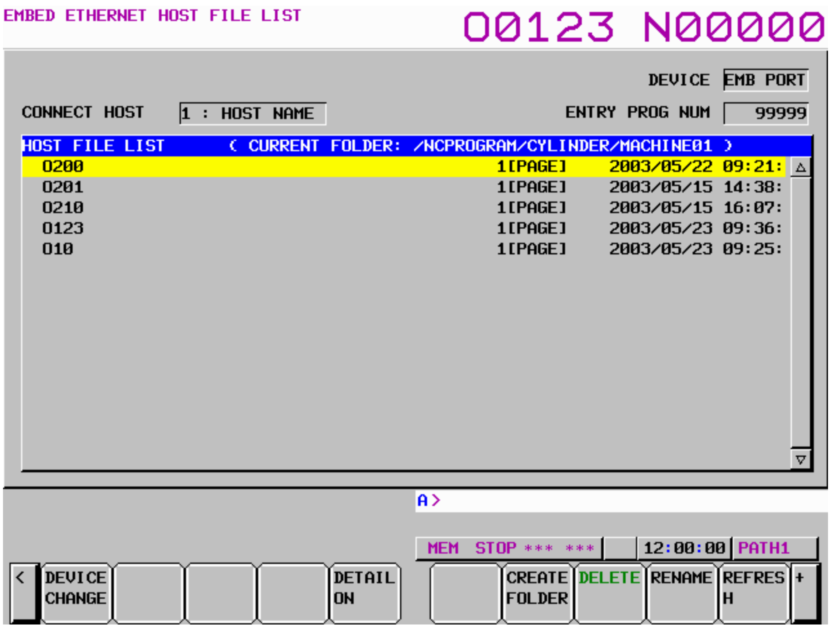


Fig. 8.3.1 (a) Embedded Ethernet host file list screen

> **NOTE**
> 1 When using the FTP file transfer function, check that the valid device is the embedded Ethernet port. The two conditions below determine a connection destination on the host file list screen:
> (1)Check that the valid device is the embedded Ethernet port. Make a selection in "DEVICE SELECTION" on the Ethernet setting screen.
> (2)A host computer can be selected from connection destinations 1, 2, and 3. Make a selection according to the [HOST] soft key described later.
> 2 File names including kanji, hiragana, and katakana characters cannot be displayed correctly.

5 When a list of files is larger than one page, the screen display can be switched using the page keys [PAGE ↑] [PAGE ↓].
6 Press the [REFRESH] soft key to update the screen display.
7 Press the [DETAIL OFF] soft key to display the host file list (file names only) screen. Press the [DETAIL ON] to display the host file list (detail) screen.
8 When selecting a folder with cursor and pressing the MDI key [INPUT] , the selected folder is changed to the new current folder.

Display item

DEVICE

"EMBED" or "PCMCIA" is displayed.

CONNECT HOST

The currently selected connection host number and host name are displayed.

ENTRY PROG NUM

The number of files registered in the work folder of the connected host is displayed. Up to 8 digits can be displayed.

CURRENT FOLDER

The current folder name of the connected host is displayed.
If the folder-path is long compared with the display-item, characters: "…" and only the last ten letters of the folder name are displayed.

FILE LIST

There is no distinction between file names and folder names.
Although the maximum number of displayed characters is 127, characters are displayed as far as they can be displayed in one line.

Operation list

DETAIL ON, DETAIL OFF

The screen display can be switched between the display of file names only and the display of details.

REFRESH

Display data can be updated.

READ

A file can be input from the host computer to the part program storage memory of the CNC. This item is displayed only when 9 is set as the I/O device number of the CNC.

PUNCH

A file can be output from the part program storage memory of the CNC to the host computer. This item is displayed only when 9 is set as the I/O device number of the CNC.

DELETE FILE

A file can be deleted from the host computer.

RENAME

A file or folder on the host computer can be renamed.

DELETE FOLDER

A folder can be deleted from the host computer.

CREATE FOLDER

A folder can be created on the host computer.

HOST

The connected host can be changed.

NC program input

A file (NC program) stored on the host computer can be input into the part program storage memory.

Procedure

1 Display the embedded Ethernet host file list screen.
2 Place the CNC in the EDIT mode.
3 Select a file to be input, with the cursor.
4 Press the [READ] soft key.
5 Press the [EXEC] soft key.
6 During input, "INPUT" blinks in the lower-right corner of the screen.

NC program output

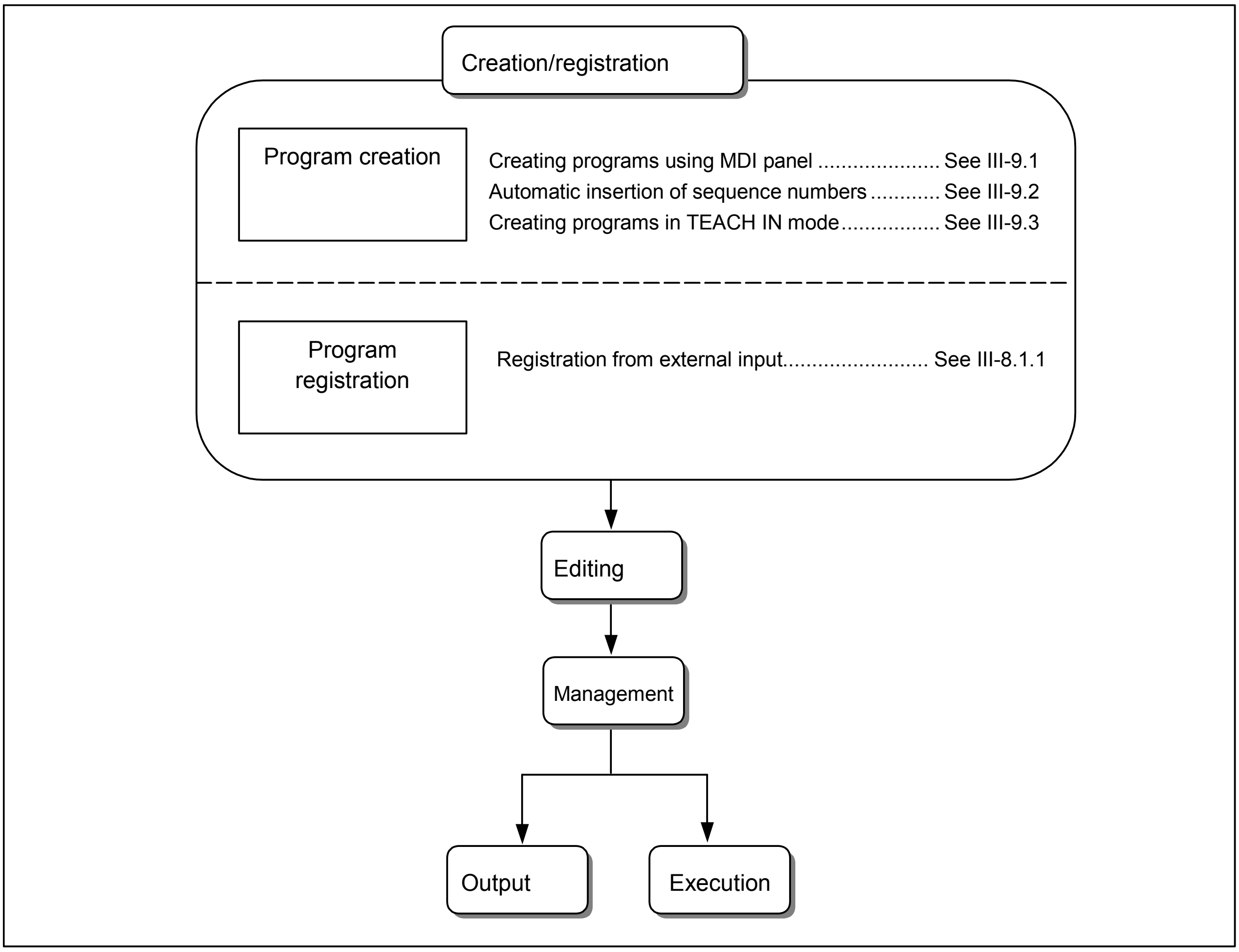
A file (NC program) stored in the part program storage memory can be output to the host computer.

Procedure

1 Display the embedded Ethernet host file list screen.
2 Place the CNC in the EDIT mode.
3 Through the MDI keyboard, enter an NC program to be output.
4 Press the [PUNCH] soft key.
5 Press the [EXEC] soft key.
6 During output, "OUTPUT" blinks in the lower-right corner of the screen.

9 CREATING PROGRAMS

This chapter explains how to create programs by MDI of the CNC.
This chapter also explains automatic insertion of sequence numbers.



9.1 CREATING PROGRAMS USING THE MDI PANEL

Programs can be created in the EDIT mode using the program editing functions described in III-10.

Procedure for Creating Programs Using the MDI Panel

1. Enter the EDIT mode.
2. Press the PROG key.
3. Press address key O and enter the program number.
4. Press the INSERT key.
5. Create a program using the program editing functions described in III-10.

Explanation

- Comments in a program

Comments can be written in a program using the control in/out codes.

Example) O0001 (TEST PROGRAM) ;
M08 (COOLANT ON) ;

- When the INSERT key is pressed after the control-out code "(", comments, and control-in code ")" have been typed, the typed comments are registered.
- When the INSERT key is pressed midway through comments, to enter the rest of comments later, the data typed before the INSERT key is pressed may not be correctly registered (not entered, modified, or lost) because the data is subject to an entry check which is performed in normal editing.

  Note the following to enter a comment:
  - Control-in code ")" cannot be registered by itself.
  - Comments entered after the INSERT key is pressed must not begin with a number, space, or address O.
  - If an abbreviation for a macro is entered, the abbreviation is converted into a macro word and registered (see Section III-10.7).
  - Address O and subsequent numbers, or a space can be entered but are omitted when registered.

9.2 AUTOMATIC INSERTION OF SEQUENCE NUMBERS

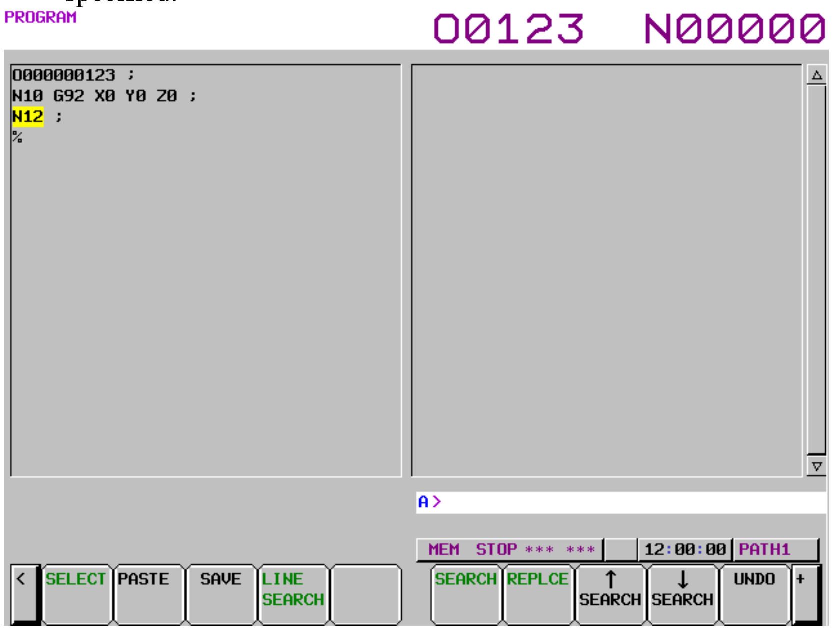
Sequence numbers can be automatically inserted in each block when a program is created using the MDI keys in the EDIT mode.
Set the increment for sequence numbers in parameter No. 3216.

Procedure for automatic insertion of sequence numbers

Procedure

1 Enter 1 in response to the prompt for "SEQUENCE NO." in setting data. (see III-12.3.1).
2 Enter the EDIT mode.
3 Press PROG to display the program screen.
4 Search for or register the number of a program to be edited and move the cursor to the EOB (;) of the block after which automatic insertion of sequence numbers is started.
When a program number is registered and an EOB (;) is entered with the INSERT key, sequence numbers are automatically inserted starting with 0. Change the initial value, if required, according to step 10, then skip to step 7.
5 Press address key N and enter the initial value of N.
6 Press INSERT .
7 Enter each word of a block.
8 Press EOB .

9 Press INSERT. The EOB is registered in memory and sequence numbers are automatically inserted. For example, if the initial value of N is 10 and the parameter for the increment is set to 2, N12 inserted and displayed below the line where a new block is specified.



10 • In the example above, if N12 is not necessary in the next block, pressing the DELETE key after N12 is displayed deletes N12.

• To insert N100 in the next block instead of N12, enter N100 and press ALTER after N12 is displayed. N100 is registered and initial value is changed to 100.

9.3 CREATING PROGRAMS IN TEACH IN MODE (PLAYBACK)

In the TEACH IN JOG or TEACH IN HANDLE mode, you can create a program while inserting the coordinate of the current position along each axis in the absolute coordinate system when the tool is moved by manual operation into the program.
You can input the words other than axis names in the same way as in the EDIT mode.

Program screen in the TEACH IN JOG mode

Displayed items

In the TEACH IN JOG or TEACH IN HANDLE MODE, the following program screen is displayed.
On the left of the screen, the coordinates of the current position in the absolute and relative coordinate systems are displayed; on the right of the screen, the contents of a program are displayed. You can create a program while checking the current position by manual operation.

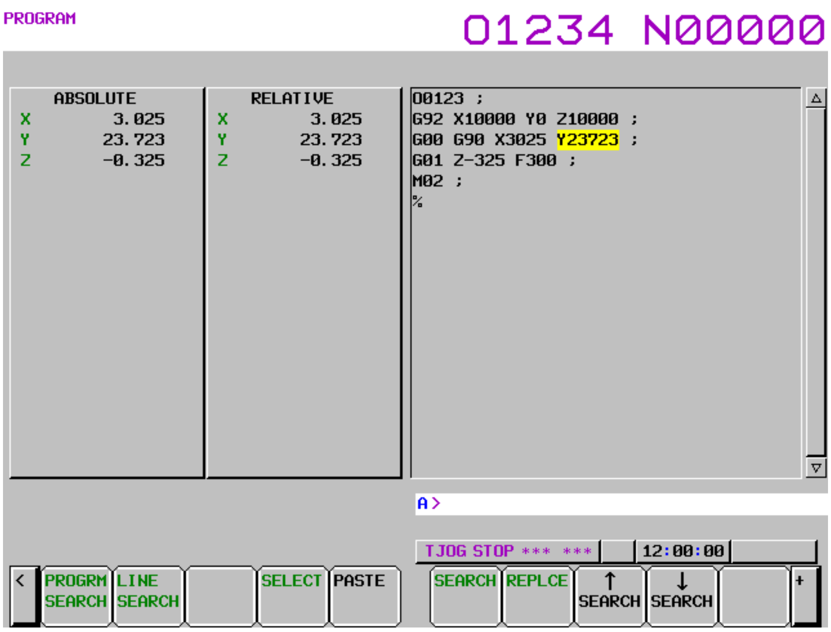


Fig. 9.3 (a) Program screen in the TEACH IN JOG mode

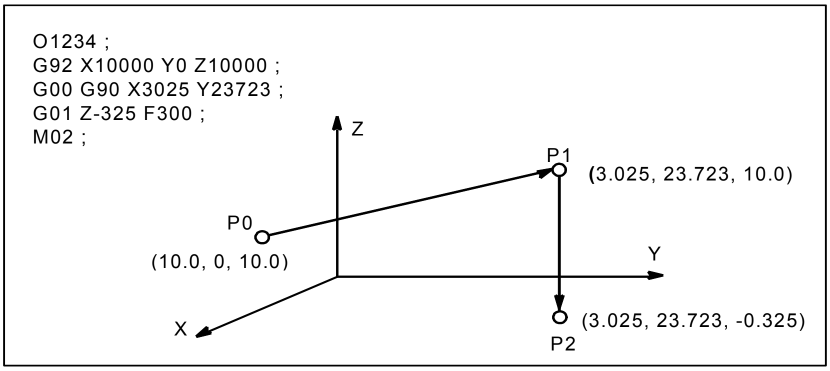
Inputting the coordinates of the current position

You can use the following procedure to insert the coordinate of the current position along each axis in the absolute coordinate system:

1 Select the TEACH IN JOG mode or TEACH IN HANDLE mode.
2 Press PROG key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the current position along each axis is to be inserted.
3 Move the tool to the desired position with jog or handle.
4 Key in the axis name of an axis along which you want to insert the coordinate of the current position.
5 Press the INSERT key. Then, the coordinate of the current position along the specified axis is inserted in the program.

(Example) X10.521 Coordinate of the current position
X10521 Data inserted in the program

Example



1 Select the TEACH IN HANDLE mode.
2 Make positioning at position P0 by the manual pulse generator.
3 Select the program screen.
4 Enter program number O1234 as follows:

O 1 2 3 4 INSERT

This operation input program number O1234 in memory.
Next, press the following keys:

EOB INSERT

An EOB (;) is entered after program number O1234.
5 Enter the P0 machine position for data of the first block as follows:

G 9 2 INSERT X INSERT Y INSERT
Z INSERT EOB INSERT

This operation registers G92X10000Y0Z10000; in program.
6 Position the tool at P1 with the manual pulse generator.

7 Enter the P1 machine position for data of the second block as follows:

G 0 0 G 9 0 INSERT

X INSERT Y INSERT EOB INSERT

This operation input G00G90X3025Z23723; in program.

8 Position the tool at P2 with the manual pulse generator.

9 Enter the P2 machine position for data of the third block as follows:

G 0 1 INSERT Z INSERT F 3 0 0 INSERT

EOB INSERT

This operation input G01Z -325F300; in program.

10 Input M02; in program as follows:

M 0 2 INSERT EOB INSERT

This completes the registration of the sample program.

Explanation

- Registering a position with compensation

When an axis name and a numeric value are keyed in and the INSERT key is pressed, the value keyed in is added to the absolute coordinate of the current position and the coordinate is inserted. This operation allows insertion of a corrected value for the absolute coordinate position.

When an extended axis name is used and the name ends with a numeric value, insert an equal sign (=) between the axis name and numeric value.

- Registering commands other than position commands

Commands to be entered before and after a position command must be entered before and after the machine position is registered, by using the same operation as program editing in EDIT mode.

- Calculator-type input

When the calculator-type input format is disabled (bit 0 (DPI) of parameter No. 3401 is set to 0), the coordinate of the current position is inserted into the program in least input increments. When the calculator-type input format is enabled (the bit is set to 1), the coordinate is inserted with a decimal point.

(Example)

Coordinate of the current position X10.521

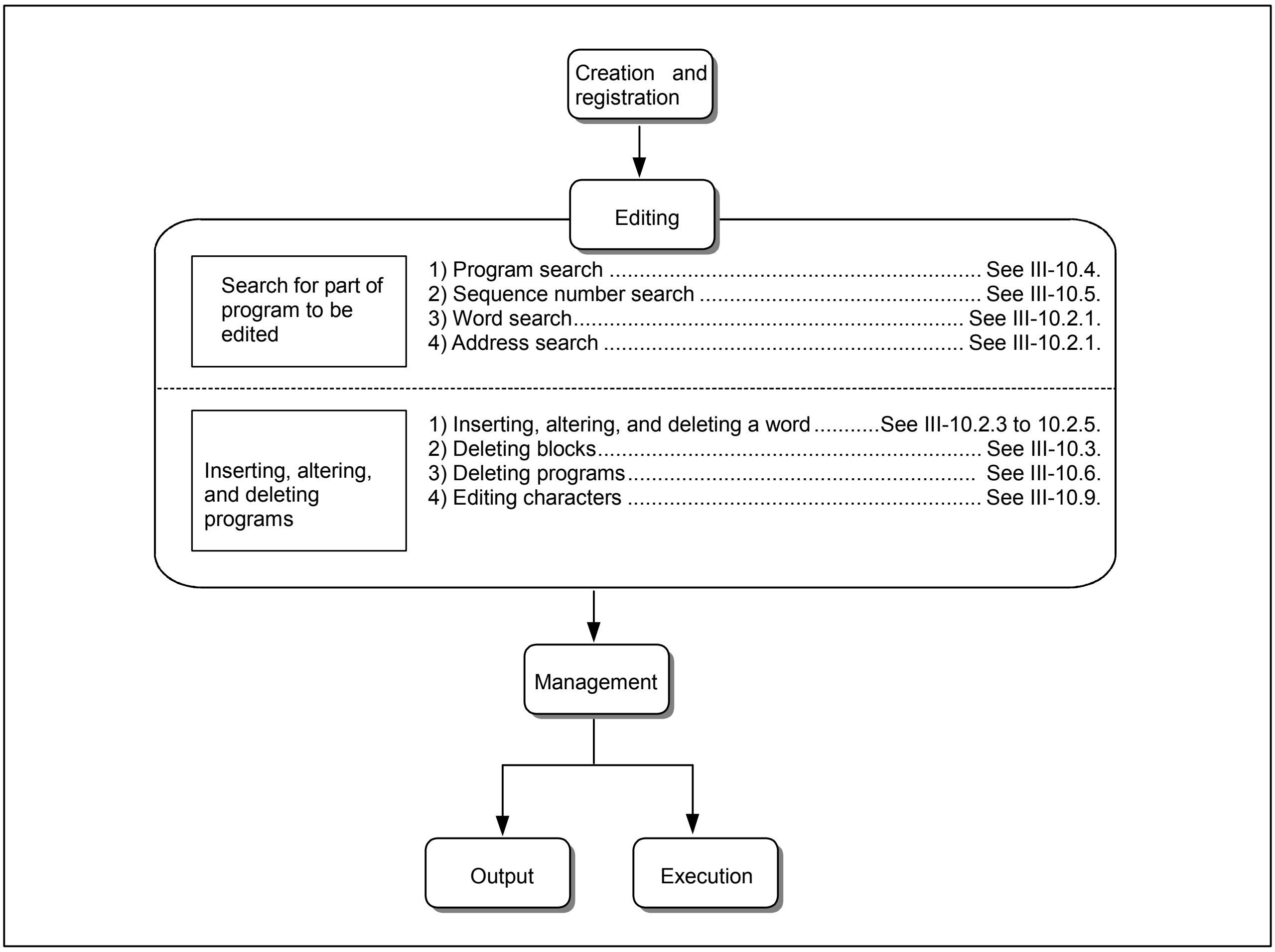
At this time, the X-axis coordinate is inserted into the program as follows:

When the calculator-type input format is disabled X10521

When the calculator-type input format is enabled X10.521

10 EDITING PROGRAMS

This chapter describes how to edit programs registered in the CNC. Editing includes the insertion, modification, and deletion of words. Editing also includes deletion of the entire program and automatic insertion of sequence numbers. In addition, PC-like program text copy and move operations are possible. This chapter also describes program search, sequence number search, word search, and address search, which are performed before editing the program.



10.1 EDIT DISABLE ATTRIBUTE

Before a program can be edited, the edit disable attribute must be removed.
The edit disable attribute can be set for each program and folder.
Programs with the edit disable attribute and programs in folders with the edit disable attribute cannot be edited.

Procedure for removing the edit disable attribute

1 Select EDIT mode.
2 Press the PROG function key.
3 Move the cursor to the program or folder from which you want to remove the edit disable attribute.
4 Press the [EDIT ENABLE] soft key.

CAUTION

1 After completing editing, set the edit disable attribute as necessary.
2 To set the edit disable attribute, follow the same procedure as for removing the attribute.
In the last step, press the [EDIT DISABL] soft key.

10.2 INSERTING, ALTERING AND DELETING A WORD

This section outlines the procedure for inserting, altering, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

1 Select EDIT mode.
2 Press function key PROG .
3 Select a program to be edited.
If a program to be edited is selected, perform the operation 4.
If a program to be edited is not selected, search for the program number.
4 Search for a word to be modified.
- Scan method
- Word search method
5 Perform an operation such as altering, inserting, or deleting a word.

Explanation

- Concept of word and editing unit

A word is an address followed by a number. With a custom macro, the concept of word is ambiguous.
So the editing unit is considered here.
The editing unit is a unit subject to alteration or deletion in one operation. In one scan operation, the cursor indicates the start of an editing unit.
An insertion is made after an editing unit.
Definition of editing unit
- Program portion from an address to immediately before the next address
- An address is an alphabet, IF, WHILE, GOTO, END, DO=, or ; (EOB).

According to this definition, a word is an editing unit.
The word "word," when used in the description of editing, means an editing unit according to the precise definition.

> ⚠ **WARNING**
> When a change, insertion, or deletion was performed on data of a program by pausing machining with the single block stop, feed hold, or other operations during execution of a program, be sure to return the cursor to the original position before restarting the program.
> To execute the program with the cursor positioned at another position, be sure to make a reset.
> Otherwise, the program may not be executed as expected from the program shown on the screen after machining restarts.

10.2.1 Word Search

A word can be searched for by merely moving the cursor through the text (scanning), by word search, or by address search.

Procedure for scanning a program

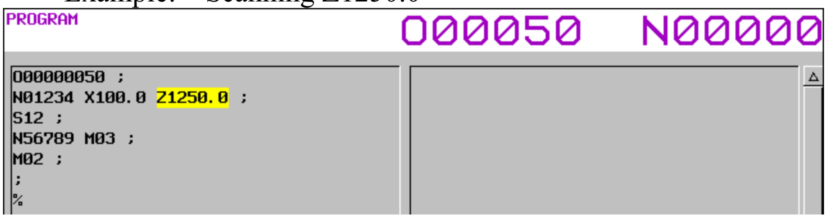
1 Press the cursor key [→].

The cursor moves forward word by word on the screen; the cursor is displayed at a selected word.

2 Press the cursor key [←].

The cursor moves backward word by word on the screen; the cursor is displayed at a selected word.

Example: Scanning Z1250.0



3 Holding down the cursor key [→] or [←] scans words continuously.

4 The first word of the next block is searched for when the cursor key [↓] is pressed.

5 The first word of the previous block is searched for when the cursor key [↑] is pressed.

6 Holding down the cursor key [↓] or [↑] moves the cursor to the head of a block continuously.

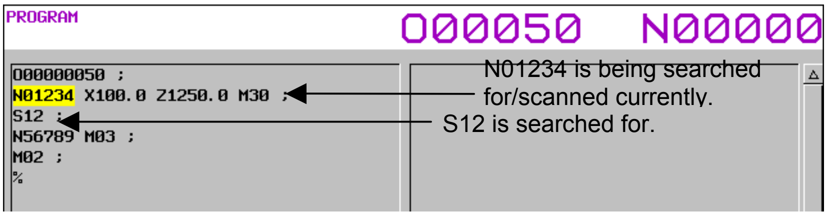
7 Pressing the page key [PAGE ↓] displays the next page and searches for the first word of the page.

8 Pressing the page key [↑ PAGE] displays the previous page and searches for the first word of the page.

9 Holding down the page key [PAGE ↓] or [↑ PAGE] displays one page after another.

Procedure for searching a word

Example of searching for S12



1 Press soft key [SRCH].

2 Key in address S .

3 Key in 1 2 .

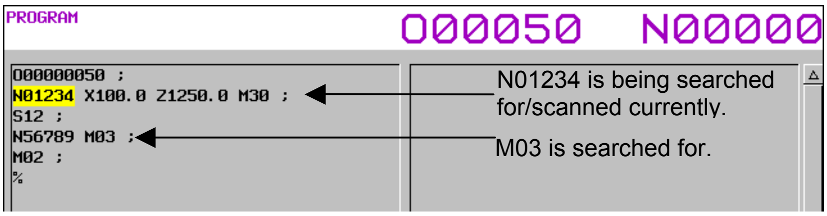
- S12 cannot be searched for if only S1 is keyed in.
- S09 cannot be searched for by keying in only S9.
  To search for S09, be sure to key in S09.

4 Pressing soft key [↓ SEARCH] starts searching forward from the cursor position. Pressing [↑ SEARCH] starts searching backward.

5 To search for the same word successively, press [↓ SEARCH] or [↑ SEARCH].

Procedure for searching an address

Example of searching for M03



1 Pressing the [SRCH] key

2 Key in address M .

3 Pressing soft key [↓ SEARCH] starts searching forward from the cursor position. Pressing [↑ SEARCH] starts searching backward.

4 To search for the same word successively, press [↓ SEARCH] or [↑ SEARCH].

10.2.2 Heading a Program

The cursor can be jumped to the top of a program. This function is called heading the program pointer. This section describes the four methods for heading the program pointer.

Procedure for heading a program

Method 1

1 Press RESET when the program screen is selected in EDIT mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2

1 Search for the program number.
When the program screen is selected in MEMORY or EDIT mode, enter a program number or program name.
When entering a program number, press address key O then type the program number.
2 Press soft key [PROGRMSEARCH].

Method 3

1 Select the program screen or program check screen in MEMORY mode.
2 Press soft key [(OPRT)].
3 Press soft key [REWIND].

Method 4

1 Select the program screen in EDIT mode.
2 Press soft key [(OPRT)].
3 Press soft key [LINE SEARCH].
4 Press soft key [TOP].

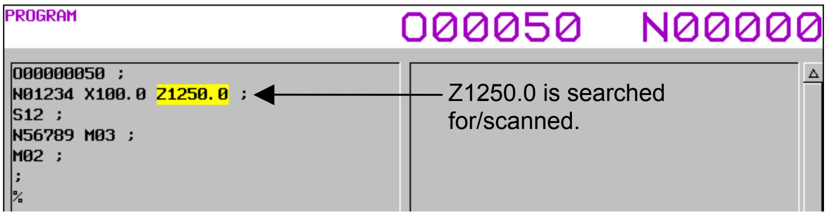
10.2.3 Inserting a Word

Procedure for inserting a word

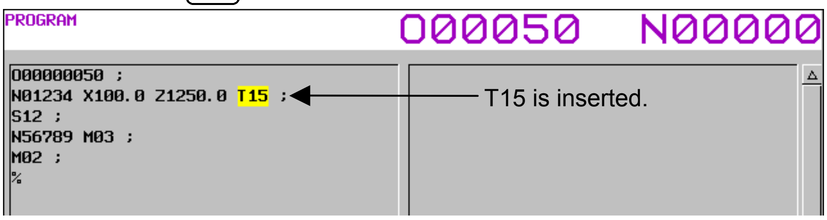
1. Search for or scan the word immediately before a word to be inserted.
2. Key in an address to be inserted.
3. Key in data.
4. Press the INSERT key.

Example of Inserting T15

1. Search for or scan Z1250.



2. Key in T 1 5 .
3. Press the INSERT key.



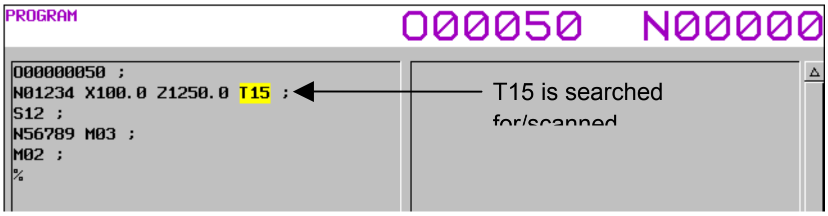
10.2.4 Altering a Word

Procedure for altering a word

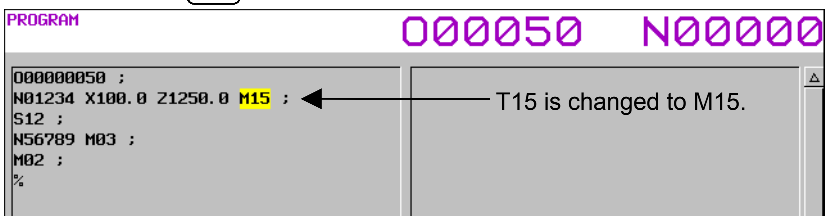
1. Search for or scan a word to be altered.
2. Key in an address to be inserted.
3. Key in data.
4. Press the ALTER key.

Example of changing T15 to M15

1. Search for or scan T15.



2. Key in M 1 5 .
3. Press the ALTER key.



10.2.5 Deleting a Word

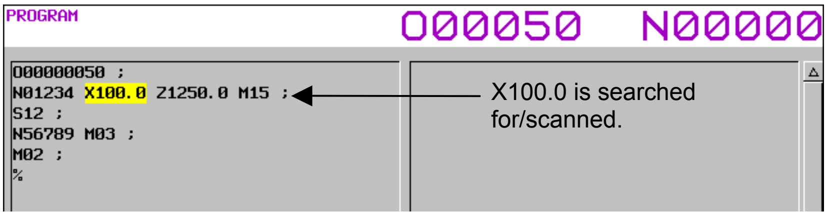
Procedure for deleting a word

1 Search for or scan a word to be deleted.

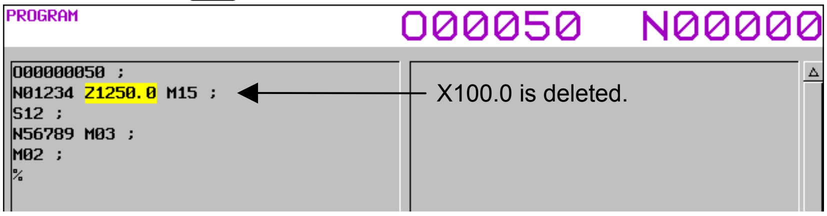
2 Press the DELETE key.

Example of deleting X100.0

1 Search for or scan X100.0.



2 Press the DELETE key.



10.3 DELETING BLOCKS

A block or blocks can be deleted in a program.

10.3.1 Deleting a Block

The portion from the current word position to the next EOB is deleted. The cursor is then placed in the word next to the deleted EOB.

Procedure for deleting a block

1 Search for or scan address N for a block to be deleted.

2 Press the [EOB] key.

3 Press the editing key [DELETE].

Example of deleting a block of N01234

1 Search for or scan N01234.

```
O00000050 ;
N01234 Z1250.0 M15 ;
S12 ;
N05678 M03 ;
M02 ;
%
```

N01234 is searched for/scanned.

2 Press the [EOB] key.

3 Press the editing key [DELETE].

```
O00000050 ;
S12 ;
N05678 M03 ;
M02 ;
%
```

Block containing N01234 has been deleted.

10.3.2 Deleting Multiple Blocks

The several blocks in the forward direction from the current word position up to the EOB of the farthest of those blocks are deleted. The cursor is then placed in the word next to the deleted EOB.

Procedure for deleting blocks

1. Search for or scan a word in the first block of a portion to be deleted.
2. Press the EOB key as many times as the number of blocks that you want to delete.
3. Press the editing key DELETE.

Example of deleting blocks from N01234 to the EOB of a block which is two blocks ahead

1. Search for or scan N01234.

```
O00000050 ;
N01234 Z1250.0 M15 ;
S12 ;
N05678 M03 ;
M02 ;
%
```

N01234 is searched for/scanned.

2. Press EOB EOB.

3. Press the editing key DELETE.

```
O00000050 ;
N5678 M03 ;
M02 ;
%
```

Blocks from N01234 to the EOB of a block which is two blocks ahead are deleted

10.4 PROGRAM SEARCH

When memory holds multiple programs, a program can be searched for. There are three methods as follows.

Procedure for program search

Method 1

1 Select EDIT or MEMORY mode.
2 Press PROG to display the program screen.
3 Enter a program number or program name. When entering a program number, press address key O then type the program number.
4 Press soft key [PROGRMSEARCH].
6 Upon completion of search operation, the program number searched for is displayed in the upper-right corner of the screen
If the program is not found , alarm PS0071 occurs.

Method 2

1 Select EDIT or MEMORY mode.
2 Press PROG to display the program screen.
3 Press soft key [PROGRMSEARCH].
4 Then, press soft key [PREV PROGRM] or [NEXT PROGRM].
Pressing [PREV PROGRM] searches for the previous program on the folder, and pressing [NEXT PROGRM] searches for the next program.

Method 3

This method searches for the program number (0001 to 0031) corresponding to a signal on the machine tool side to start automatic operation. Refer to the relevant manual prepared by the machine tool builder for detailed information on operation.

1 Select MEMORY mode.
2 Set the reset state.
  - The reset state is the state where the LED for indicating that automatic operation is in progress is off. (Refer to the relevant manual of the machine tool builder.)
3 Set the program number selection signal on the machine tool side to a number from 01 to 31.
4 Press the cycle start button.
  - When the signal on the machine tool side represents 00, program number search operation is not performed.
  - If the program corresponding to a signal on the machine tool side is not registered, alarm DS0059 is raised.

10.5 SEQUENCE NUMBER SEARCH

Sequence number search operation is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number.

Example)

Sequence number 02346 in a program (O0002) is searched for.

PROGRAM

O0001 ;
N01234 X100.0 Z100.0 ;
S12 ;
:
Selected program → ***O0002 ;***
N02345 X20.0 Z20.0 ;
Target sequence number is found. → ***N02346 X10.0 Z10.0 ;***
:
O0003 ;
:

This section is searched starting at the beginning. (Search operation is performed only within a program.)

Procedure for sequence number search

Procedure

1 Select MEMORY mode.

2 Press ROG PROG .

3 If the program contains a sequence number to be searched for, perform the operations 4 to 7 below.
If the program does not contain a sequence number to be searched for, select the program number of the program that contains the sequence number to be searched for.

4 Key in address N .

5 Key in a sequence number to be searched for.

6 Press soft key [N SRH].

7 Upon completion of search operation, the sequence number searched for is displayed in the upper-right corner of the screen.
If the specified sequence number is not found in the program currently selected, alarm PS0060 occurs.

Explanation

- Operation during Search

Those blocks that are skipped do not affect the CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values.
So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify M, S, and T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and NC states at that point.

- Checking during search

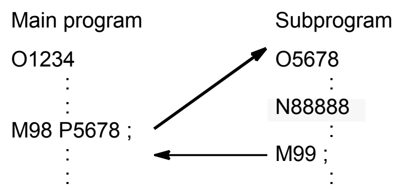
During search operation, the following checks are made:

- Optional block skip

Limitation

- Searching in sub-program

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm PS0060 is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.



If an attempt is made to search for N8888 in the example above, an alarm occurs.

Fig. 10.5 (a)

10.6 DELETING PROGRAMS

Programs registered in memory can be deleted, either one program by one program or all at once.

10.6.1 Deleting One Program

A program in the default folder is deleted.

Procedure for deleting one program

1 Select the EDIT mode.
2 Press function key PROG to display the program screen.
3 Key in address O .
4 Key in a desired program number.
5 Press the editing DELETE key. The program with the entered program number is deleted.

10.6.2 Deleting All Programs

All programs in the default folder are deleted.

Procedure for deleting all programs

1 Select the EDIT mode.
2 Press function key PROG to display the program screen.
3 Key in address O .
4 Key in -9999.
5 Press editing key DELETE to delete all programs.

10.7 EDITING OF CUSTOM MACROS

Unlike ordinary programs, custom macro programs are modified, inserted, or deleted based on editing units.
Custom macro words can be entered in abbreviated form.
Comments can be entered in a program.
Refer to the III-9.1 for the comments of a program.

Explanation

- Editing unit

When editing a custom macro already entered, the user can move the cursor to each editing unit that starts with any of the following characters and symbols:

(a) Address
(b) # located at the start of the left side of a substitution statement
(c) /, (,=, and ;
(d) First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT and PCLOS

On the screen, a blank is placed before each of the above characters and symbols.
Example) Head positions where the cursor is placed

```
N001 X-#100 ;
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X-SQRT[#3/3*[#4+1]] ;
N004 X-#2 Z#1 ;
N005 #5 =1+2-#10 ;
IF[#1NE0] GOTO10 ;
WHILE[#2LE5] DO1 ;
#[200+#2] =#2*10 ;
#2 =#2+1 ;
END1 ;
```

- Abbreviations of custom macro word

When a custom macro word is altered or inserted, the first two characters or more can replace the entire word.
Namely,

WHILE → WH	GOTO → GO	XOR → XO	AND → AN
SIN → SI	ASIN → AS	COS → CO	ACOS → AC
TAN → TA	ATAN → AT	SQRT → SQ	ABS → AB
BCD → BC	BIN → BI	FIX → FI	FUP → FU
ROUND → RO	END → EN	POPEN → PO	BPRNT → BP
DPRNT → DP	PCLOS → PC	EXP → EX	THEN → TH

Example) Keying in

WH [AB [#2] LE RO [#3]] has the same effect as
WHILE [ABS [#2] LE ROUND [#3]]
The program is also displayed in this way.

10.8 PASSWORD FUNCTION

The password function locks bit 4 (NE9) of parameter No. 3202, which protects programs with program Nos. O9000 to O9999 and programs and folders having the edit/display disable attribute, according to the settings in two parameters, PASSWORD (parameter No. 3210) and KEYWD (parameter No. 3211). In the locked state, parameter NE9 cannot be set to 0. In this case, the protection of programs with program Nos. O9000 to O9999 and programs and folders having the edit/display disable attribute cannot be released unless a correct keyword is set.
A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

Procedure for locking and unlocking

Locking

1 Set the MDI mode.
2 Enable parameter writing (III-12.3.1). At this time, alarm PS0100 is issued on the CNC.
3 Set parameter No. 3210 (PASSWD). At this time, the locked state is set.
4 Disable parameter writing.
5 Press the RESET key to release the alarm state.

Unlocking

1 Set the MDI mode.
2 Enable parameter writing (III-12.3.1). At this time, alarm PS0100 is issued on the CNC.
3 In parameter No. 3211 (KEYWD), set the same value as set in parameter No. 3210 (PASSWD) for locking. At this time, the locked state is released.
4 Set parameter NE9 (No. 3202#4) to 0.
5 Disable parameter writing.
6 Press the RESET key to release the alarm state.
7 Subprograms from program Nos. 9000 to 9999 can now be edited.

Explanation

- Setting parameter PASSWD

The locked state is set when a value is set in the parameter PASSWD. However, note that parameter PASSWD can be set only when the locked state is not set (when PASSWD = 0, or PASSWD = KEYWD). If an attempt is made to set parameter PASSWD in other cases, a warning is given to indicate that writing is disabled. When the locked state is set (when PASSWD ≠ 0 and PASSWD ≠ KEYWD), parameter NE9 is automatically set to 1. If an attempt is made to set NE9 to 0, a warning is given to indicate that writing is disabled.

- Changing parameter PASSWD

Parameter PASSWD can be changed when the locked state is released (when PASSWD = 0, or PASSWD = KEYWD). After step 3 in the procedure for unlocking, a new value can be set in the parameter PASSWD. From that time on, this new value must be set in parameter KEYWD to release the locked state.

- Setting 0 in parameter PASSWD

When 0 is set in the parameter PASSWD, the number 0 is displayed, and the password function is disabled. In other words, the password function can be disabled by either not setting parameter PASSWD at all, or by setting 0 in parameter PASSWD after step 3 of the procedure for unlocking. To ensure that the locked state is not entered, care must be taken not to set a value other than 0 in parameter PASSWD.

- Re-locking

After the locked state has been released, it can be set again by setting a different value in parameter PASSWD, or by turning the power to the NC off then on again to reset parameter KEYWD.

⚠ CAUTION

1 Once the locked state is set, parameter NE9 cannot be set to 0 and parameter PASSWD cannot be changed until the locked state is released or the memory all-clear operation is performed. Special care must be taken in setting parameter PASSWD.
2 The edit/display disable attribute cannot be set unless parameter PASSWD is set.
3 In the locked state, programs with the edit/display disable attribute are treated as follows:
   - The presence of the programs is hidden. This means that these programs are not displayed on screens such as the program directory screen. These programs cannot be edited either.
   - These programs cannot be selected as a main program. They can be called as subprograms.
4 In the unlocked state, programs with the edit/display disable attribute are treated in the same manner as ordinary programs.
5 The programs in a folder having the edit/display disable attribute are also treated as described in Caution 3 and 4 above.
6 In the locked state, the folders in a folder having the edit/display disable attribute are treated as follows:
   - The presence of the folders is hidden. This means that these folders are not displayed on screens such as the program directory screen.
7 In the unlocked state, the folders in a folder having the edit/display disable attribute are treated in the same manner as ordinary folders.

10.9 EDITING PROGRAM CHARACTERS

This section describes how to edit programs registered in the CNC. Editing operations include character insertion, modification, deletion, and replacement.

While program word editing is performed by recognizing program words, program character editing is performed on a character-by-character basis.

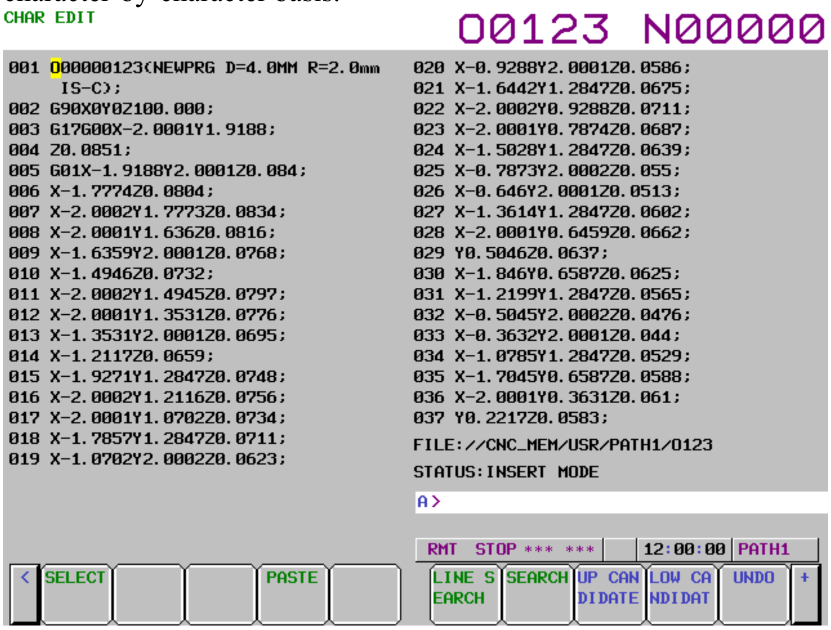


Fig. 10.9 (a)

Explanation

- Edit unit

Character editing is performed on a character-by-character basis. Select either character editing or word editing according to the editing type or experience.

- Line

A line is defined as a range containing a character string and a terminal symbol ";".

From the edited line, a line is used as a unit for reading and writing.

When one program line contains many characters, it extends over multiple lines on the screen, but these lines are counted as one program line.

- Line splitting

If edit key INPUT is entered when the cursor is placed in the middle of a line during line editing, the characters before the cursor and the characters at the cursor position and subsequent positions are treated in separate lines.
To restore the original single line, press edit key CANCEL immediately after the line is split.
Alternatively, move the cursor to the terminal symbol ";" of the first line, and press edit key DELETE to restore the original single line.

- Line merging

When the terminal symbol ";" at the end of a line is deleted, that line and the next line are merged into a single line.

- Maximum number of characters in one line

One line can consist of up to 140 characters.

- Line number

The number of lines are counted starting with the starting line of a program, which is counted as the first line. Even when a line wraps around to the next and subsequent lines, these lines are counted as a single line.

- Clipboard

The clipboard is an area to store characters when a cut or copy operation is performed. The area has a capacity for holding approximately 4000 characters.
The characters that are cut or copied by one cut or copy operation are stored. The most recently stored character string can be used by the paste function.
The information stored by the paste operation is kept unchanged until another cut or copy operation is performed.
The characters stored in the clipboard are maintained until the power to the CNC is turned off unless the clipboard is updated again.

- Undo function

The undo function in text editing restores the state present before each edit operation by reversing operations in time from the most recent operation. Only functions for updating text are reversed.
One undo operation corresponds to one input operation.
When characters are input, one input operation corresponds to an operation on one character.
When text has been updated by replacement operations, one undo operation reverses one replacement operation.
When text has been updated using the replace all function, one undo operation reverses one replacement operation.
The move keys (cursor keys →, ←, ↑, and ↓, and page keys PAGE ↓ and ↑ PAGE) do not update text, so key operations of these keys are not reversed by the undo function.

- Example of the undo function

1 Suppose that the following character string is present before modifications:
   N110AX[#AXIS3]=100.0;
2 Edit key [DELETE] is pressed five times.
   X[#AXIS3]=100.0;
3 Soft key [UNDO] is pressed five times. Then, the original character string is restored as follows:
   AX[#AXIS3]=100.0;
   0AX[#AXIS3]=100.0;
   10AX[#AXIS3]=100.0;
   110AX[#AXIS3]=100.0;
   N110AX[#AXIS3]=100.0;

- Edit mode

There are two program edit modes: insert mode and overwrite mode.
Program editing is performed in one of these modes.
To switch between edit modes, use soft key [MODE].
Initially, insert mode is set.

- Insert mode

In insert mode, an entered character is inserted between the current cursor position and the preceding character position.

1234567890

When the cursor is placed at 6, and X is entered, the following results:

12345X67890

- Overwrite mode

In overwrite mode, the character at the cursor position is replaced by an entered character.

1234567890

When the cursor is placed at 6, and X is entered, the following results:

12345X7890

- Restrictions on editing

O numbers and file names cannot be edited.
EOR (%) cannot be deleted.

- Line update and automatic saving

When a line has been updated, the line is indicated in the update color. When the cursor moves outside the updated line, the color of the line changes to the non-update color.
When the cursor has moved outside the updated line, the line is saved automatically.

- Relationship between automatic saving of updated lines and the undo function

Because updated lines are automatically saved, text restored by the undo function is automatically saved.
When a certain line is updated, then the cursor is moved to the next line, the updated line is written by the automatic saving function.
Then, using the undo function restores the original state. In this case, the updated line has already been written by the automatic saving function, so writing is performed to restore the original state.

10.9.1 Available Keys

The available keys are as follows:

- Cursor keys

Cursor keys →, ←, ↑, and ↓ move the cursor.

- Editing key DELETE

Deletes the character at the cursor position.

- Editing key CANCEL

Deletes the character immediately before the cursor position.
When the cursor is at the beginning of a line, the character at the end of the previous line is deleted.

- Editing key INPUT

Causes a line change.

- Page change keys

Pages are changed using page key PAGE ↓ or ↑ PAGE.
Page key PAGE ↓ moves to the next page.
When page key PAGE ↓ is pressed on the last page, the cursor moves to the last character position of the last line (the % position).
Page key ↑ PAGE moves to the previous page.
When page key ↑ PAGE is pressed on the first page, the cursor moves to the first character position of the first line.
Page key PAGE ↓ moves to the next page.
When page key PAGE ↓ is pressed on the last page, the cursor moves to the last character position of the last line (the % position).

- Data keys

Characters usable in a program can be entered.

10.9.2 Input Mode

Input modes include insert mode and overwrite mode.

Changing input mode

To change input mode, use soft key [INSERT MODE] or [OVERWRITE MODE].
Pressing soft key [OVERWRITE MODE] enters overwrite mode if the current mode is insert mode.
Pressing soft key [INSERT MODE] enters insert mode if the current mode is overwrite mode.
The current mode is indicated in the lower right part of the editing screen.

10.9.3 Line Number Display

This function is used to display a program with line numbers. Pressing soft key [LINE NUMBER] displays a program with line numbers. A second press of soft key [LINE NUMBER] displays the program with no line numbers. Line numbers are added to a program when the program is displayed, but they are not written to files.

10.9.4 Search

A program is searched for a character string.

Search

Procedure

1. Press soft key [SEARCH].
2. A character string input/edit area for search appears. Enter the character string to be searched for.
3. Upward search operation
   When soft key [UP] is pressed, the program is searched upward (toward the top) from the current cursor position.
   If the search character string is found, the cursor is placed on the character string.
   When soft key [UP] is pressed again, the program is searched for the next candidate.
4. Downward search operation
   When soft key [DOWN] is pressed, the program is searched downward (toward the EOR) from the current cursor position.
   If the search character string is found, the cursor is placed on the character string.
   When soft key [DOWN] is pressed again, the program is searched for the next candidate.
   When the character string to be searched for is no longer found, the cursor returns to the original position.

10.9.5 Replacement

A character string in a program is replaced with a specified character string.

Replacement

Procedure

1 To replace a character string, press soft key [ALTER].

2 A replacement character string input/edit area appears. Enter the character string (the search character string) that you want to search for and the character string (the replacement character string) with which you want to replace the search character string.
To move between the search character string input area and replacement character string input area, use cursor keys [⬇] and [⬆].
In each input area, the cursor position before the movement operation is restored.
When a search character string is entered without entering a replacement character string, the specified character string is deleted.

3 Replacement operation
The replacement operation involves searching for a character string and replacing the character string.
For searching for a replacement character string, soft keys [↑ SEARCH] and [↓ SEARCH] are provided.
For replacement, [REPLACE] and [REPLACEALL] are provided.

Soft key [↑ SEARCH]
The program is searched upward (toward the beginning of the program) from the current cursor position. If the search character string is found, the cursor is placed on the character string.
When soft key [↑ SEARCH] is pressed again, the program is searched for the next candidate.

Soft key [↓ SEARCH]
The program is searched downward (toward the EOR) from the current cursor position.
If the search character string is found, the cursor is placed on the character string.
When soft key [↓ SEARCH] is pressed again, the program is searched for the next candidate.

Soft key [REPLACE]
Pressing this key replaces the character string searched for with the replacement character string.

Soft key [REPLCEALL]
This key performs replacements throughout the program text at a time.
When this key is pressed, message "Do you want to execute?" appears together with soft keys [YES] and [NO]. If soft key [YES] is pressed, all replacements are made.
If soft key [NO] is pressed, the original editing screen is redisplayed without performing replacement.

10.9.6 Reversing Edit Operations (Undo Function)

This function reverses edit operations.
Edit operations made on a program are reversed in time from the most recent operation.

Reversing edit operations (undo function)

Procedure

1 Each time soft key [UNDO] is pressed, one operation is reversed.
A write to a file is performed.
When there is no operation to be reversed, pressing this soft key is ignored.
Even when there is a difference between the records indicated at the time of the operation and the currently indicated records, there is no problem.

10.9.7 Selection

A character string to be copied or deleted is selected.

Selection

Procedure

1 Place the cursor at the beginning of a target character string.
2 Press soft key [SELECT].
3 Then, move the cursor to the end of the target range by using cursor keys [→], [←], [↑], and [↓], and page keys [PAGE ↑] and [PAGE ↓].
The selected character string is then displayed in the selection color (the background color is the cursor color).
4 Press soft key [COPY] or [CUT].
After soft key [COPY] or [CUT] is pressed, the normal cursor state is restored.
To cancel the selection, press soft key [CANCEL]. Then, the screen display is returned to the editing screen.

10.9.8 Copy

A selected character string is stored in the clipboard.
The text on the screen remains unchanged.

Copying

Procedure

1 Specify the character string to be copied by following the selection procedure described previously.
2 Press soft key [COPY].

10.9.9 Deletion

A selected character string is saved in the clipboard, and the character string selected on the screen is deleted.

Deletion

Procedure

1 Specify the character string to be cut by following the selection procedure described previously.
2 Press soft key [DELETE].

10.9.10 Paste

The character string in the clipboard is inserted to the current cursor position.

Paste

Procedure

1 Move the cursor to the position at which you want to paste a character string.
2 Press the soft key [PASTE].

10.9.11 Saving

When editing ends, the text portion not yet written is saved by pressing soft key [SAVE].
Press [SAVE] when completing editing.

Saving

Procedure

1 Press the soft key [SAVE].

10.9.12 Creation

A program to be edited is displayed on the screen.

Creation

Procedure

1 Press soft key [CREATE].
2 A program name input area appears.
3 Enter the name of a program to be created.
4 Press the soft key [EXEC]. This creates a new program and displays the editing screen.
To stop this operation halfway, press soft key [CANCEL].

10.9.13 Line Number Search

The cursor moves to a specified line number.

Searching for a line number

Procedure

1 Press soft key [LINE SEARCH].
2 A line number input area appears.
3 Enter the line number to which you want to move. Line numbers start with 1.
4 Press the soft key [LINE NUMBER]. This moves the cursor to the specified line number.
5 To move to the beginning of the program, press the soft key [LINE NUMBER].

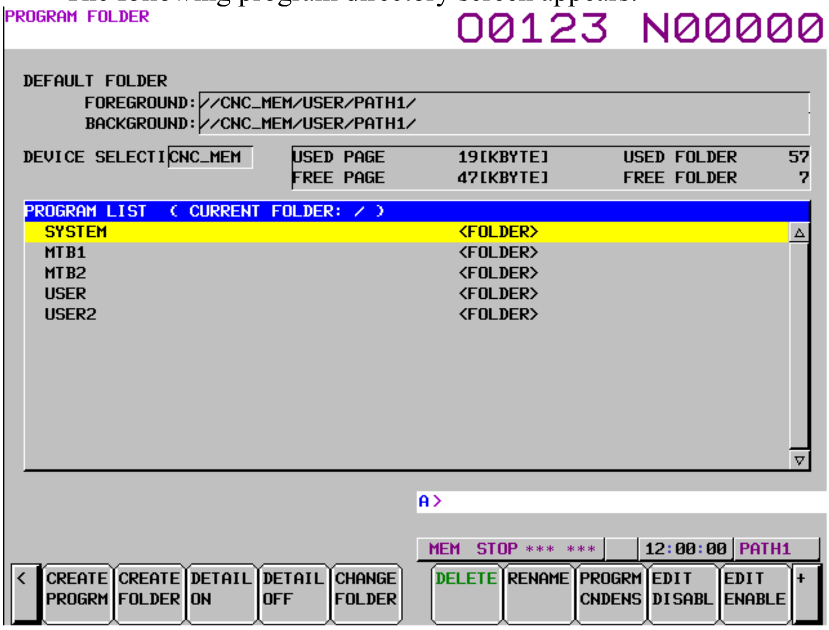
10.10 PROGRAM COPY FUNCTION

A program is copied or moved between folders.

Procedure

1 Press function key PROG .

2 Press chapter selection soft key [DIRECTORY].
The following program directory screen appears:



3 Press the soft key [(OPRT)].

4 Move the cursor to a folder that contains the program that you want to copy or move, and press soft key [CHANGE FOLDER].

5 Press continuous menu key several times until [SELECTSTART] appears.

6 Press the soft key [SELECTSTART].

7 Move the cursor to the program that you want to select, and press soft key [SELECT]. The background color of the selected program becomes light blue, indicating the selected state.

8 Press the soft key [SELECT END]. The selected program is determined.

9 Press soft key [CHANGE FOLDER], and move the cursor to the folder to which you want to copy or move the program.

10 Press the soft key [COPY]. The selected program is copied. If just one program is selected, pressing soft key [COPY] after typing a program name performs a copy operation with the entered name.

11 To move the program, press soft key [MOVE]. The selected file is then moved. If just one program is selected, pressing soft key [MOVE] after typing a program name performs a copy operation with the entered name.

Explanation

Operations are accepted only when the data protection key is set to ON.

If the program storage capacity on the copy destination side is insufficient, the copy operation is not accepted.

The currently selected program is highlighted. Multiple programs can be selected in the same folder. Each time [SELECT] is pressed, the program currently indicated by the cursor is selected.

A selected program can be deselected by pressing [SELECT] again or by pressing [CANCEL].

If the same program name is already present in the copy or move destination folder, "OVERWRITE: FILE NAME" is displayed, allowing you to determine whether to overwrite the existing program by pressing soft key [NO] or [OVERWRITE]. Pressing [OVERWRITE] performs overwriting, and pressing [NO] does not perform the copy or move operation of that program.

Pressing soft key [DELETE] deletes all the files currently selected.

A program can be neither copied nor moved to the same folder as the selected folder.
When only one program is selected, and a program name is already entered, however, the program can be copied or moved within the same folder.

NOTE

Once a copy or move operation starts, it cannot be canceled. So, start the operation very carefully.

10.11 KEYS AND PROGRAM ENCRYPTION

Overview

Program contents can be protected by setting parameters for encryption and for the program security range.

Explanation

1 Security with a password and a security range
When the password and security range parameters are specified, the display, editing, and input/output operations are disabled for the programs within the security range.
This prevents the custom macro programs created by the machine tool builder from being accidentally changed or deleted by the end user. This also provides security because the contents of the programs are not displayed.

2 Input/output of encrypted programs
The programs in the security range can be encrypted before being output. Once encrypted, the programs cannot be decrypted. In addition, encrypted programs can be input directly.

- Locking/unlocking

When the programs in the security range are secured, the program memory is said to be locked.
If they are not secured, the program memory is said to be unlocked.

- Parameter

- Parameter PASSWORD (No.3220)
  Set the password required to lock the program memory. A password other than 0 can be set. The password is not displayed.
  A password can be set when no password is set (PASSWORD = 0) or the program memory is unlocked.
- Parameter KEY (No.3221)
  When KEY is set equal to PASSWORD, the program memory is unlocked. The value set for the parameter is never displayed. Each time the CNC is started, 0 is set for the parameter. This means that whenever the CNC is started, a lock is provided if a password is set (PASSWORD $\neq$ 0).
- Minimum value (in parameter No. 3222) and maximum value (in parameter No. 3223) of the program security range
  Set the desired security range. The value set for MINIMUM must not be greater than that set for MAXIMUM. The programs in the range of MINIMUM to MAXIMUM are secured. If MINIMUM is set to 0, it is regarded as being 9000. If MAXIMUM is set to 0, it is regarded as being 9999.
  A range can be set when no password is set or the program memory is unlocked.

> **NOTE**
> 1 For security, the values set for PASSWORD and KEY are not displayed. For the same reason, PASSWORD, MINIMUM, and MAXIMUM can be specified only when no password is set or the program memory is unlocked. Set a password, taking great care to avoid a situation where the program memory cannot be unlocked because a password is set incorrectly.
> 2 The [+INPUT] key used to specify PASSWORD and KEY behave in the same way as the [INPUT] soft key.
> Example: When 99 is set for KEY, pressing 1 and [+INPUT] changes the value to.
> 3 These four parameters cannot be output externally. Also, even when these parameters are input by the parameter read operation, they are ignored.

- Inputting/outputting and collating programs

When a program is encrypted, a password is output. The password is used to load the program.
The following output operations are enabled for programs outside the security range if the locked state is present or for programs within the security range if the unlocked state is present:

Outputting all programs

Locked/unlocked	Results
Locked	All the programs outside the security range are output in the normal way.
Unlocked	All the programs within the security range are encrypted and output.
Password not set	All the programs in the program memory are output in the normal way.

Outputting a single program

Locked/unlocked	Results
Locked	If a program is outside the security range, it is output in the normal way. If it is within the security range, a warning "PROGRAM NOT FOUND" is issued.
Unlocked	If a program is outside the security range, it is output in the normal way. If it is within the security range, it is encrypted and output.
Password not set	The program is output in a normal way.

Outputting specified multiple programs

Locked/unlocked	Results
Locked	When all of the specified programs fall outside the protected range, they are output as usual. When all of the specified programs are within the security range, warning message "PROGRAM NOT FOUND" is issued. When some of the specified program numbers are outside the security range, and the others are within the security range, only the programs outside the security range are output normally. When the programs to be output are not within the specified range, warning message "PROGRAM NOT FOUND" is issued.
Unlocked	When all of the specified programs fall outside the protected range, they are output as usual. When all the specified programs fall within the protected range, they are output as encoded programs. When some of the specified program numbers are outside the security range, and the others are within the security range, only the programs within the security range are encoded and output. When the programs to be output are not within the specified range, warning message "PROGRAM NOT FOUND" is issued.
Password not set	The program is output in a normal way.

Inputting an un-encrypted program

Locked/unlocked	Results
Locked	When the program to be read is outside the security range, it is input normally. When the program to be read is within the security range, warning message "WRITE PROTECTED" is issued.
Unlocked, or password not set	The program is input.

Inputting an encrypted program

Password set in the system and password of the program	Results
Password set in the system	Warning “WRITE PROTECT” is issued.
Password for the program	When the program is within the security range, it is input normally. When the program is outside the security range, warning message "WRITE PROTECTED" is issued.
Password not set in the system	The program is input. The PSW in the file is set for parameter No. 3220.

Collating a program with an encrypted program

In the unlocked state, the following takes place:

Password set in the system and password of the program	Results
Password set in the system	Alarm SR0075 “PROTECT” is issued.
Password set in the system = Password of the program, or password not set in the system	The program is collated.

In the locked state, program collation cannot be performed.

NOTE
1 To encrypt programs, set parameter ISO (bit 1 of No.0000) to 1 (to specify that the punch code is ISO).
2 An encrypted program cannot be registered as an additional program ([READ]-[ADD]).

- Program display

1 On the program directory screen, all program numbers are displayed together with comments.

2 In the locked state, the programs within the security range are not displayed on the program screen. In the unlocked state, the programs within the security range are displayed in the same way as normal programs.

- Editing and deleting programs

When the program memory is locked, the programs within the security range cannot be edited or deleted. When the program memory is locked, an attempt to delete all programs results in only those programs outside the security range being deleted.

- Searching for programs

In the locked state, a program search is performed within the protected range as described below.

1 When no program number is specified, programs within the protected range are skipped.

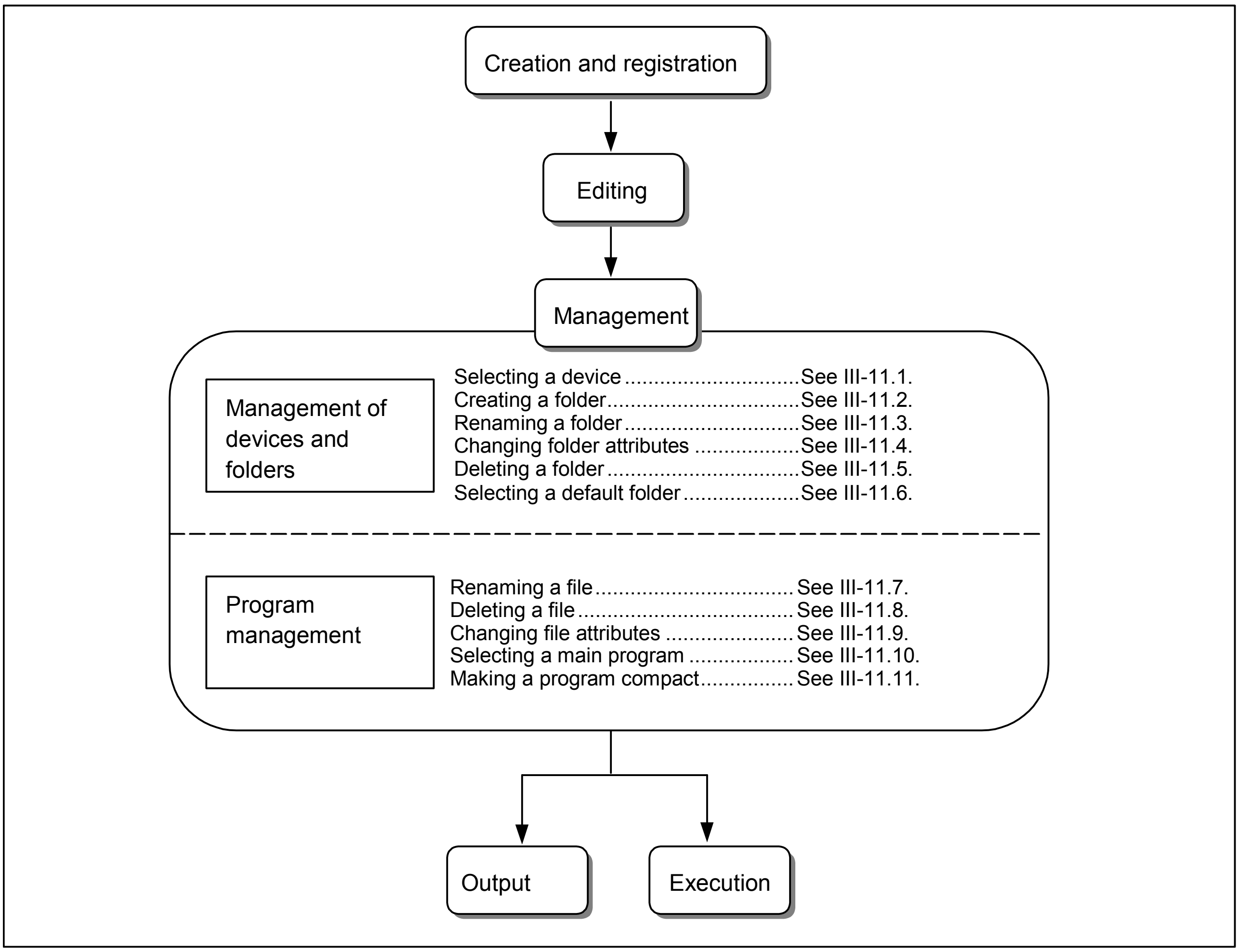
2 When an attempt is made to search for a program within the security range by specifying a program number, the search operation is ignored, resulting in warning message "PROGRAM NOT FOUND".

11 PROGRAM MANAGEMENT

Program management functions are classified into the following two types:

- Functions for folders
- Functions for programs

Functions for folders include creation, deletion, change of names and attributes, and so on.
Functions for programs include main program selection, deletion, change of names and attributes, program compaction, and so on.



11.1 SELECTING A DEVICE

When the fast data server function (option) is provided, a program storage device can be selected. This section explains the selection procedure.

Procedure for selecting a device

1 Press the function key PROG .
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key for the desired device.

11.1.1 Selecting a Memory Card Program as a Device

Overview

By selecting a memory card including a program storage file (named "FANUCPRG.BIN") as a device, memory operation can be performed with the program in the program storage file selected as the main program.
In addition, the content of a program storage file can be displayed on the program list screen or a program in a program storage file can be edited on the program edit screen.
A program storage file can be created using a memory card program tool (A08B-9010-J700#ZZ11) on a commercially available personal computer. To use a created program storage file, the file is written to a memory card prepared in the FAT format.
(A program held in a program storage file is hereinafter referred to as a memory card program. Moreover, a memory card storing a program storage file is referred to as a program storage memory card.)

Procedure for selecting a device

1 Press the function key PROG .
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key [MEMCARD].

NOTE
1 A FAT-formatted memory card containing the program storage file FANUCPRG.BIN is recognized as a program storage memory card.
2 For a program storage memory card containing more than 63 folders and programs, the option for extending the number of memory card program registrations is required. The option for extending the number of memory card program registrations is applicable to a program storage memory card containing a maximum of 1000 folders and programs.

Procedure for removing a device

When a program storage memory card is replaced or a memory card is used for normal usage such as data input/output, clear the recognition of the program storage memory card with removal operation.

1 Press the function key PROG .
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key [DETACH].

> **NOTE**
> 1 This soft key appears when the CNC recognizes a program storage memory card during device change operation.
> 2 This operation is enabled only in EDIT mode or MEM mode.
> When a memory card program is selected in the main programs of multiple paths in a multipath control system, set the modes of all paths to EDIT mode or MEM mode.
> 3 When the default folder is a folder in a program storage file, it is changed to "//CNC_MEM/" by a removal operation.
> 4 When the main program is a memory card program, the main program enters the unselected state by a removal operation.

Explanation

- About operation

A memory card program can be selected as a main program to perform memory operation.
Memory operation has the following features:

- Subprogram call nesting is allowed.
- Macro program call nesting is allowed.
- In a custom macro, a control command using a GOTO statement/WHILE statement can be specified.
- With the T series, a multiple repetitive canned cycle for turning can be specified.

> **NOTE**
> To use the functions for calling macros, using custom macros, and using a multiple repetitive canned cycle for turning, the options corresponding to the respective functions are required.

- Selection as a main program

As a main program to be automatically executed in the memory mode, a memory card program can be selected.

- Sub program (call using M98/G72.1/G72.2)
- Macro program (call using G65/G66/G66.1/M96)

The following subprogram/macro program held in the same folder as containing the main program is called:

- Sub program call (M98)
- Macro call (Simple macro call G65 / Modal call G66,G66.1)
- Macro interrupt (M96)
- Figure copying (G72.1, G72.2)

If the program cannot be found in the same folder as the main program, the following folder is searched:

- Common program folder on the CNC_MEM device (CNC program storage memory)

> **NOTE**
> To use the functions for macro call, macro interrupt, and figure copying, the options corresponding to the respective functions are required.

- Sub program (call using M code/S code/T code/particular address/the second auxiliary function)
- Macro program (call using G code/M code)

The following subprogram/macro program calls a program from the CNC_MEM device (CNC program storage memory):

- Subprogram call using M code/S code/T code/particular addresses/the second auxiliary function
- Macro call using G code/M code

The folders set as search targets are searched in the following order, and the first found program is called:

1 Common program folder among the initial folders
2 MTB-dedicated folder 2 among the initial folders
3 MTB-dedicated folder 1 among the initial folders
4 System folder among the initial folders

Search target folders are set in parameter No. 3457.

> **NOTE**
> For a memory card program, subprogram call using M code/S code/T code/particular addresses/the second auxiliary function or macro call using G code/M code can be specified. However, a program on the CNC_MEM device (CNC program storage memory) is called.

- External program number search / External workpiece number search

A program on a program storage memory card can be searched for with the external program number search function or external workpiece number search function.

Limitation

For a memory card program, M198 cannot be specified. Moreover, no memory card program can be called from a program on the CNC_MEM device (CNC program storage memory) by specifying M198.

When a setting is made to enable an external device subprogram call from a memory card (M198) or DNC operation from a memory card (bit 7 (MNC) of parameter No. 0138 = 1), the content of program storage file cannot be displayed during automatic operation.

When a program storage memory card is selected, the memory card cannot be used for the ordinary purposes listed below. To use a memory card in such a case, perform a "removal" operation to cancel the recognition of the program storage memory card.

- ALL I/O screen
  Display of the contents of a memory card, and reading/punching data to and from a memory card
- PMC data I/O screen
  Display of the contents of a memory card, and reading/punching to and from a memory card
- Program directory screen
  Reading/punching program data to and from a memory card
- External device subprogram call (M198) operation
  Subprogram call (M198) with a memory card set as an external device
- DNC operation
  DNC operation from a memory card

⚠ CAUTION

1. Do not remove the memory card when a program that specifies a write to the memory card is being edited. The data can be destructed.
2. If an editing operation is completed, the results of editing are preserved even when the power to the CNC is turned off.
3. When removing the memory card, be sure to perform a "removal" operation. If the memory card is removed without performing a "removal" operation and an attempt is made to access the memory card, the alarm (SR1964) or alarm (IO1030) is issued.
   If the card is removed inadvertently, insert the card again and perform a "removal" operation.
   When an alarm is issued, perform the following operation:
   - When the alarm (SR1964) is issued
     Reset the alarm after performing a "removal" operation.
   - When the alarm (IO1030) is issued
     The alarm can be reset only by turning off the power to the CNC.
4. Do not attempt to replace the memory card without performing a "removal" operation. Such an attempt is very dangerous.

- Operation of creattion, edition, and management of a program

When “memory card program as a device” is selected, operation of creattion, edition, and management of a program is below:

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	Unusable
Inserting, alteration, and deletion a Word	Usable
Deletion of a block	Usable
Program search	Usable
Sequence number search	Usable
Deletion of a program	Unusable
Editing a customa macro	Usable
Password function	Unusable
Editing a program character	Usable
Program copy function	Unusable
Key and progrm encryption	Unusable
Selecting a device	Usable
Creation of a folder	Unusable
Renaming of a folder	Unusable
Alteration of folder’s attribute	Unusable
Deletion of a folder	Unusable
Selecting a default folder	Usable
Renaming of a file	Unusable
Deletion of a file	Unusable
Alteration of folder attribute	Unusable
Selecting a main program	Usable
Execution of reduce of programs	Unusable
Input/output of program	Unusable

11.2 CREATING A FOLDER

This section explains the procedure for creating a folder.

Procedure for creating a folder

1 Select EDIT mode.
2 Press the function key PROG .
3 Move to the folder in which you want to create a folder.
Use the cursor keys ⬇ and ⬆ to move among folders.
After selecting the folder, press the INPUT key.
4 Press the soft key [(OPRT)].
5 Type data.
6 Press the soft key [CREATEFOLDER].

NOTE
1 Each folder name must be unique within the same folder.
2 Each time a folder is created, the number of programs that can be registered decreases by one.
3 Depending on the operation status and protection status, a folder cannot sometimes be created.

11.3 RENAMING A FOLDER

This section explains the procedure for renaming a folder.

Procedure for renaming a folder

1 Select EDIT mode.
2 Press the function key PROG .
3 Press the soft key [FOLDER].
4 Select the folder that you want to rename.
To select a folder, use the cursor keys ⬇ and ⬆.
5 Press the soft key [(OPRT)].
6 Type data.
7 Press the soft key [RENAME].

NOTE
1 You cannot rename the initial folders.
2 Each folder name must be unique within the same folder.
3 Depending on the operation status and protection status, a folder cannot sometimes be renamed.

11.4 CHANGING FOLDER ATTRIBUTES

This section explains the procedure for changing the attribute of a folder (edit disable or edit/display disable).

Procedure for changing folder attributes

1 Select EDIT mode.
2 Press the function key PROG .
3 Press the soft key [FOLDER].
4 Select the folder of which attribute is to be changed.
To select a folder, use the cursor keys ⬇ and ⬆.
5 Press the soft key [(OPRT)].
6 Press the soft key [DETAILON].
7 Press the soft key [+].
- To disable editing, press the soft key [EDIT DISABL].
- To enable editing, press the soft key [EDIT ENABLE].
- To disable editing and display, press the soft key [DISP DISABL].
- To enable editing and display, press the soft key [DISP ENABLE].

NOTE
1 Depending on the operation status and protection status, the attribute of a folder cannot sometimes be changed.
2 When the edit disable attribute is set for a folder, editing of folders and files in that folder is disabled.
3 When the edit/display disable attribute is set for a folder, editing and display of folders and files in that folder is disabled, and these folders and files are not displayed.
4 The items that can be set vary depending on the status of parameters and so on.

11.5 DELETING A FOLDER

This section explains the procedure for deleting a folder.

Procedure for deleting a folder

1. Select EDIT mode.
2. Press the function key PROG .
3. Press the soft key [FOLDER].
4. Select the folder that you want to delete.
   To select a folder, use the cursor keys ↓ and ↑.
5. Press the soft key [(OPRT)].
6. Press the soft key [DELETE].
   - To perform the deletion, press the soft key [EXEC].
   - To cancel the deletion, press the soft key [CANCEL].

> **NOTE**
> 1. You cannot delete the initial folders.
> 2. A folder cannot be deleted unless the folder is empty.
>   (An empty folder means that the folder contains neither folders nor files.)
> 3. If a folder contains a folder or file having the edit/display disable attribute, the folder may seem to be empty when displayed, but the folder is not actually empty, so it cannot be deleted.
> 4. Depending on the operation status and protection status, a folder cannot sometimes be deleted.

11.6 SELECTING A DEFAULT FOLDER

This section explains the procedure for selecting a foreground or background default folder.

Procedure for selecting a default folder

1 Select EDIT mode.
2 Press the function key PROG .
3 Press the soft key [FOLDER].
4 Move to the folder that you want to select as the default folder.
Use the cursor keys ↓ and ↑ to move among folders.
After selecting the folder, press the INPUT key.
5 Press the soft key [(OPRT)].
6 • To select the foreground, press the soft key [CHANGE FORE].
• To select the background, press the soft key [CHANGE BACK].

NOTE
1 When the foreground or background default folder is not set, the path folder, which is an initial folder, is assumed.
2 The settings for the foreground and background default folders are stored in the default folder setting file.
3 When a program file, program folder, or program folder management file is cleared, the default folder setting file is also cleared at the same time.

11.7 RENAMING A FILE

This section explains the procedure for renaming a file.

Procedure for renaming a file

1 Select EDIT mode.
2 Press the function key PROG .
3 Press the soft key [FOLDER].
4 Move to the folder containing the file that you want to rename.
Use the cursor keys ⬇ and ⬆ to move among folders.
After selecting the folder, press the INPUT key.
5 Select the file that you want to rename.
To select a file, use the cursor keys ⬇ and ⬆.
6 Press the soft key [(OPRT)].
7 Type data.
8 Press the soft key [RENAME].

NOTE
1 Each file name must be unique within the same file.
2 When the assigned file name cannot be treated as a program number, the program in the file is restricted as follows:
   - Specification by program number is impossible. (Such as a subprogram call)
   - Information output by program number is impossible.
3 Depending on the operation status and protection status, a file cannot sometimes be renamed.

11.8 DELETING A FILE

This section explains the procedure for deleting a file.

Procedure for deleting a file

1 Select EDIT mode.
2 Press the function key PROG .
3 Press the soft key [FOLDER].
4 Move to the folder containing the file that you want to delete.
Use the cursor keys ↓ and ↑ to move among folders.
After selecting the folder, press the INPUT key.
5 Select the file that you want to delete.
To select a file, use the cursor keys ↓ and ↑.
6 Press the soft key [(OPRT)].
7 Press the soft key [DELETE].
- To perform the deletion, press the soft key [EXEC].
- To cancel the deletion, press the soft key [CANCEL].

NOTE
Depending on the operation status and protection status, a file cannot sometimes be deleted.

11.9 CHANGING FILE ATTRIBUTES

This section explains the procedure for changing the attribute of a file (edit disable, edit/display disable, encoding, or protection of data at eight levels).

Procedure for selecting the attribute of a file

1 Select EDIT mode.

2 Press the function key PROG .

3 Press the soft key [FOLDER].

4 Move to the folder containing the file of which attribute is to be changed.
Use the cursor keys ↓ and ↑ to move among folders.
After selecting the folder, press the INPUT key.

5 Select the file of which attribute is to be changed.
To select a file, use the cursor keys ↓ and ↑.

6 Press the soft key [(OPRT)].

7 Press the soft key [DETAILON].

8 Press the soft key [+].

9
- To disable editing, press the soft key [EDIT DISABL].
- To enable editing, press the soft key [EDIT ENABLE].
- To disable editing and display, press the soft key [DISP DISABL].
- To enable editing/display, press the soft key [DISP ENABLE].
- To set encoding, press the soft key [ENCODESET].
- To cancel encoding, press the soft key [ENCODE RESET].
- To change the change protection level, type a change protection level, then press the soft key [CHANGELEVEL].
- To change the output protection level, type an output protection level, then press soft key [OUT LEVEL].

NOTE

1 Depending on the operation status and protection status, a file cannot sometimes be deleted.

2 The items that can be set vary depending on the status of options, parameters, and so on.

11.10 SELECTING A MAIN PROGRAM

This section explains the procedure for selecting a main program.

Procedure for selecting a main program

1 Select EDIT mode.
2 Press the function key PROG .
Press the soft key [FOLDER].
3 Move to the folder containing the file that you want to use as a main program.
Use the cursor keys ⬇ and ⬆ to move among folders.
After selecting the folder, press the INPUT key.
4 Select the file that you want to use as the main program.
To select a file, use the cursor keys ⬇ and ⬆.
5 Press the soft key [(OPRT)].
6 Press the soft key [MAIN PROGRM].

NOTE
Depending on the operation status and protection status, the main program cannot sometimes be selected.

11.11 MAKING A PROGRAM COMPACT

This section explains the procedure for making a program compact.

Procedure for making a program compact

1 Select EDIT mode.
2 Press the function key PROG .
Press the soft key [FOLDER].
3 Move to the folder containing the file of the program that you want to make compact.
Use the cursor keys ⬇ and ⬆ to move among folders. After selecting the folder, press the INPUT key.
3 Select the file of the program that you want to make compact.
To select a file, use the cursor keys ⬇ and ⬆.
4 Press the soft key [(OPRT)].
5 Press the soft key [PROGRMCNDENS].

NOTE
1 Depending on the operation status and protection status, a program cannot sometimes be made compact.
2 nly programs on the CNC_MEM device can be made compact.

12 SETTING AND DISPLAYING DATA

To operate a CNC machine tool, various data must be set on the MDI panel for the CNC. The operator can monitor the state of operation with data displayed during operation.
This chapter describes how to display and set data for each function.

Explanation

- Screen transition chart

The screen transition for when each function key on the MDI panel is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.

See Chapter 7 for the screen that appears when function key MESSAGE is pressed. In general, function key CUSTOM1 or CUSTOM2 is prepared by the machine tool builder and used for macros. Refer to the manual issued by the machine tool builder for the screen that appears when function key CUSTOM1 or CUSTOM2 is pressed.

- Data protection key

The machine may have a data protection key to protect part programs, tool compensation values, setting data, and custom macro variables. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

Screen displayed when the function key POS is pressed

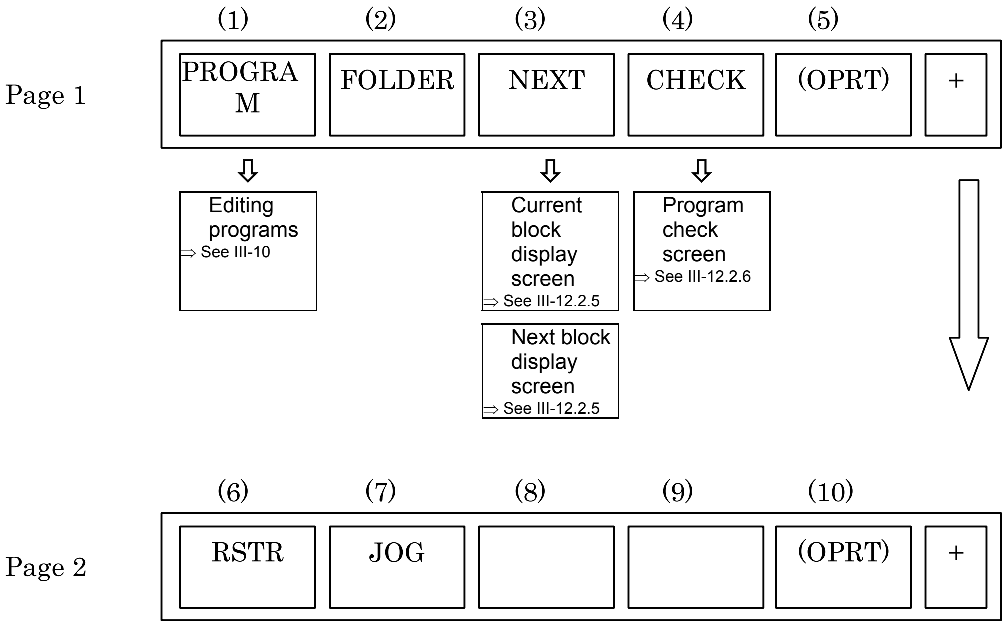
Page 1

(1)	(2)	(3)	(4)	(5)	
ABS	REL	ALL	HNDL	(OPRT)	+
Position display in the workpiece coordinate system ⇨ See III-12.1.1	Position display in the workpiece coordinate system ⇨ See III-12.1.1	Position display in the workpiece coordinate system ⇨ See III-12.1.1	Manual handle interruption ⇨ See III-4.4		
Actual feedrate display ⇨ See III-12.1.5	Actual feedrate display ⇨ See III-12.1.5	Actual feedrate display ⇨ See III-12.1.5			
Display of run time and parts count ⇨ See III-12.1.6	Display of run time and parts count ⇨ See III-12.1.6	Display of run time and parts count ⇨ See III-12.1.6			

Page 2

(6)	(7)	(8)	(9)	(10)	
MONI	5AXMAN			(OPRT)	+
Operating monitor display ⇨ See III-12.1.8	Display of manual feed for 5-axis machining ⇨ See III-12.1.9				

Screen displayed when the function key PROG is pressed



Screen displayed when the function key OFFSET SETTING is pressed

Page 1

(1)	(2)	(3)	(4)	(5)	
OFFSET	SETTING	WORK		(OPRT)	+
Setting and displaying the tool offset value ⇒ See III-2.1.1 *1	Displaying and entering setting data ⇒ See III-12.3.1	Displaying and setting the workpiece origin offset value ⇒ See III-12.3.4			

Page 2

(6)	(7)	(8)	(9)	(10)	
MACRO		OPR	TOOL MANAGER	(OPRT)	+
Displaying and setting custom macro common variables ⇒ See III-12.3.6		Displaying and setting the software operator's panel ⇒ See III-12.3.8	Setting and displaying tool management data ⇒ See III-12.3.9		

Page 3

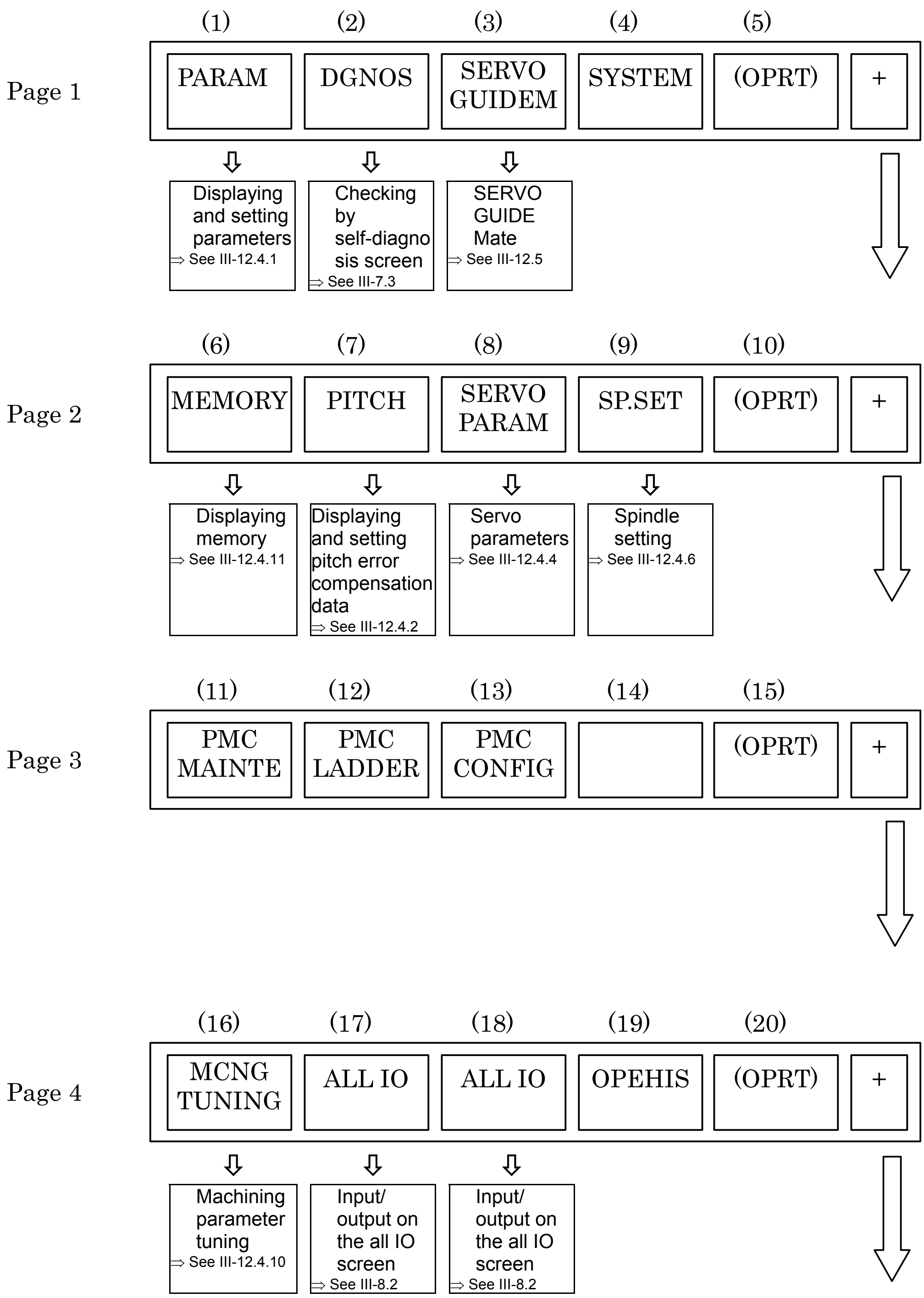
(11)	(12)	(13)	(14)	(15)	
OFST.2	W.SHFT	GEOM.2		(OPRT)	+
Y-axis offset ⇒ See III-2.1.6 *1	Setting the workpiece coordinate system shift value ⇒ See III-2.1.4 *1	Setting tool compensation / second geometry offset values ⇒ See III-2.1.5 *1			

Page 4

(16)	(17)	(18)	(19)	(20)	
	PR-LV			(OPRT)	+
	Precision level selection ⇒ See III-12.3.11				

	(21)	(22)	(23)	(24)	(25)	
Page 5	CHUCK TAIL	LANG.	PROTECT	GUARD	(OPRT)	+
	⇩ Chuck and tail stock barriers ⇨ See III-2.1.7[*1]	⇩ Displaying and switching the display language ⇨ See III-12.3.10	⇩ Protection of data at eight levels ⇨ See III-12.3.11	⇩ Operation confirmation functions ⇨ See III-6.5		

Screen displayed when the function key SYSTEM is pressed



Page 5

(21)	(22)	(23)	(24)	(25)	
COLOR	MAINTE	M-INFO	W. DGNS	(OPRT)	+

⇩ (21) Color setting screen ⇒ See III-12.4.9

Page 6

(26)	(27)	(28)	(29)	(30)	
	FSSB	PRMTUN		(OPRT)	+

⇩ (27) FSSB data display and setting screen ⇒ See Maintenance Manual

⇩ (28) Machining parameter tuning ⇒ See III-12.4.10

Page 7

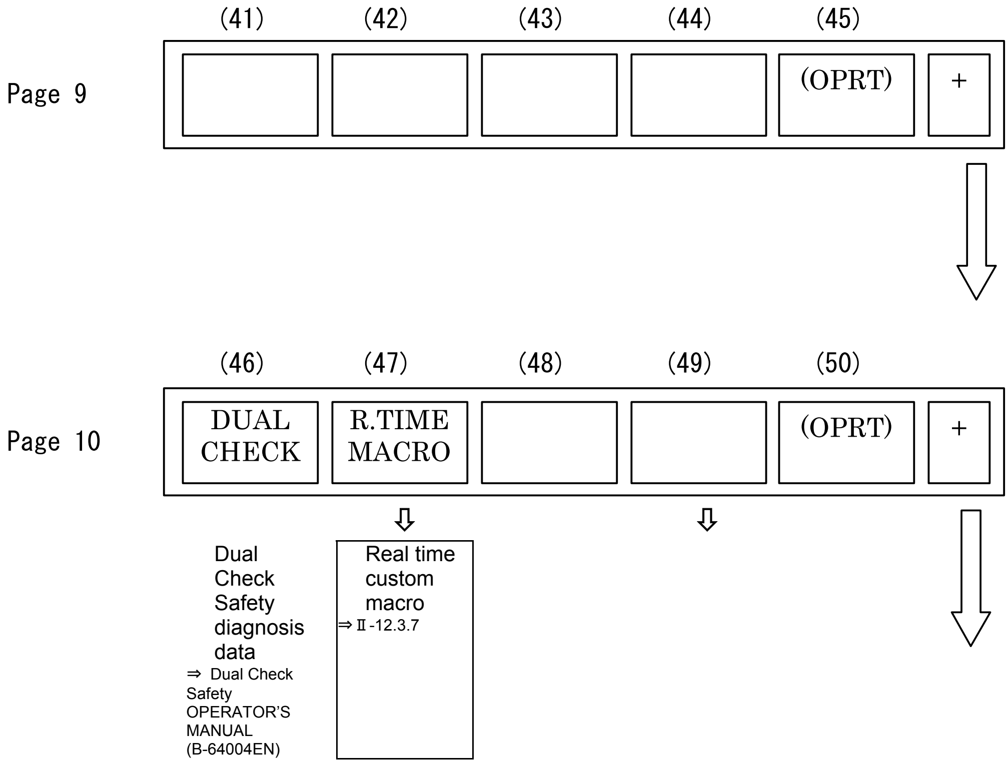
(31)	(32)	(33)	(34)	(35)	
EMBEDDED PORT	PCMCIA LAN	ETHNET BOARD	PROFI-BUS MST	(OPRT)	+

Page 8

(36)	(37)	(38)	(39)	(40)	
	M CODE		3D ERR COMP	(OPRT)	+

⇩ (37) M code grouping function ⇒ II-11.3

⇩ (39) 3-dimensional error compensation data ⇒ II-12.4.3



NOTE

For the screen dedicated to each of the path control types for the lathe system/machining center system, refer to the following manuals:

*1: User's Manual (T series) (B-63944EN-1)

*2: User's Manual (M series) (B-63944EN-2)

12.1 SCREENS DISPLAYED BY FUNCTION KEY POS

Press function key POS to display the current position of the tool.

The following three screens are used to display the current position of the tool:

- Current position display screen for the workpiece coordinate system.
- Current position display screen for the relative coordinate system.
- Current overall position display screen.

The above screens can also display the feedrate, run time, and the number of parts. In addition, a floating reference position can be set on these screens.

Function key POS can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key POS can also be used to display the screen for displaying the distance moved by handle interruption. See III- 4.4 for details on this screen.

12.1.1 Position Display in the Workpiece Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate

Procedure

1 Press function key POS .

2 Press soft key [ABS].

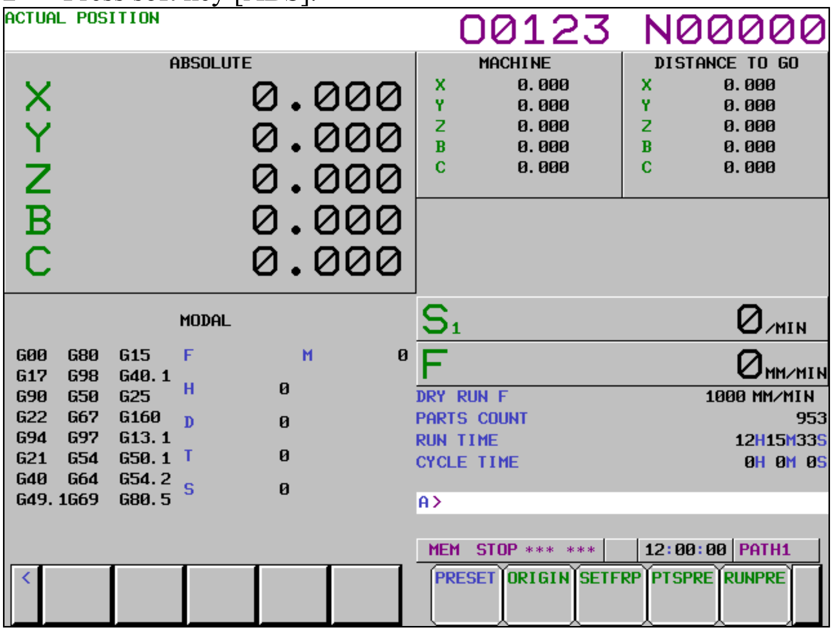


Fig. 12.1.1 (a) Current position (absolute) screen (M series)

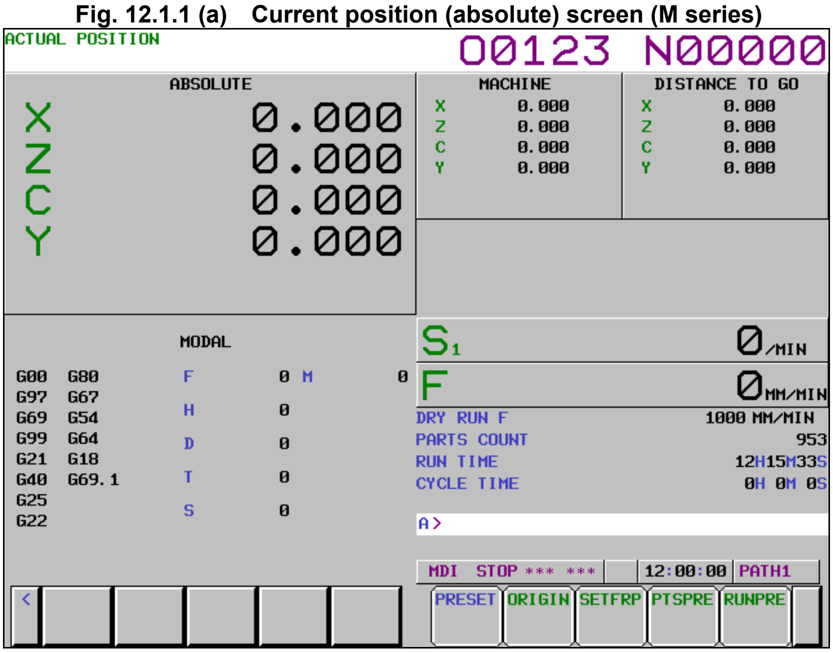


Fig. 12.1.1 (b) Current position (absolute) screen (T series)

Explanation

- Display including compensation values

M

Parameters DAL, DAC (No. 3104#6, #7) can be used to select whether the displayed values include tool length compensation and cutter compensation.

T

Parameters DAP, DAC (No. 3129#1, No.3104#7) can be used to select whether the displayed values include tool offset and tool nose radius compensation.

12.1.2 Position Display in the Relative Coordinate System

Displays the current position of the tool in a relative coordinate system based on the coordinates (see Explanation) set by the operator. The current position changes as the tool moves. The increment system is used as the unit for numeric values.
The title at the top of the screen indicates that relative coordinates are used.

Display procedure for the current position screen with the relative coordinate system

Procedure

1 Press function key POS .

2 Press soft key [REL].

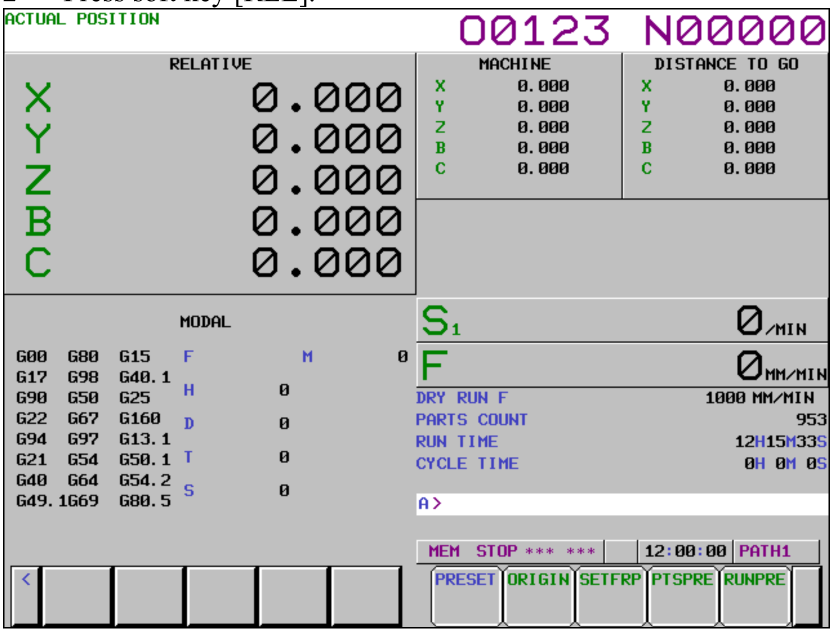


Fig. 12.1.2 (a) Current position (relative) screen (M series)

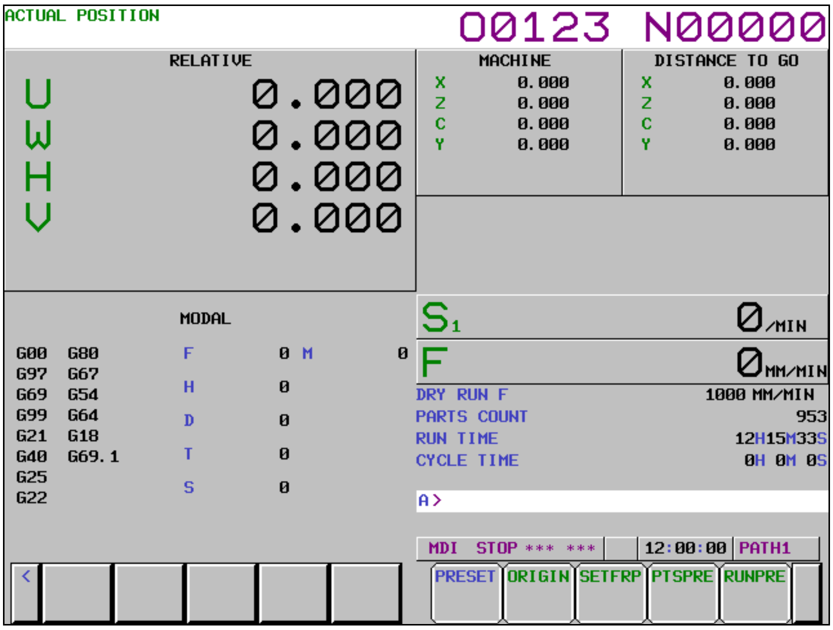


Fig. 12.1.2 (b) Current position (relative) screen (T series)

See Explanation for the procedure for setting the coordinates.

Explanation

- Setting the relative coordinates

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows:

Procedure to set the axis coordinate to a specified value

Procedure

1 To reset the coordinate to 0, press soft key [ORGIN].
Key in an axis name to be reset (such as X or Y), then press soft key [EXEC].

2 For presetting to a specified value, key in an axis name to be preset and a preset value (such as X100.000), then press soft key [PRESET].

- Display including compensation values

M

Parameters DRL, DRC (No. 3104#4, #5) can be used to select whether the displayed values include tool length compensation and cutter compensation.

T

Parameters DRP, DRC (No.3129#0, No.3104#5) can be used to select whether the displayed values include tool offset and tool nose radius compensation.

- Presetting by setting a coordinate system

M

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G92 command.

T

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G50 command (for G code system A) or G92 command (for G code system B or C).

12.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance. The relative coordinates can also be set on this screen. See III-12.1.2 for the procedure.

Procedure for displaying overall position display screen

Procedure

1 Press function key POS .

2 Press soft key [ALL].

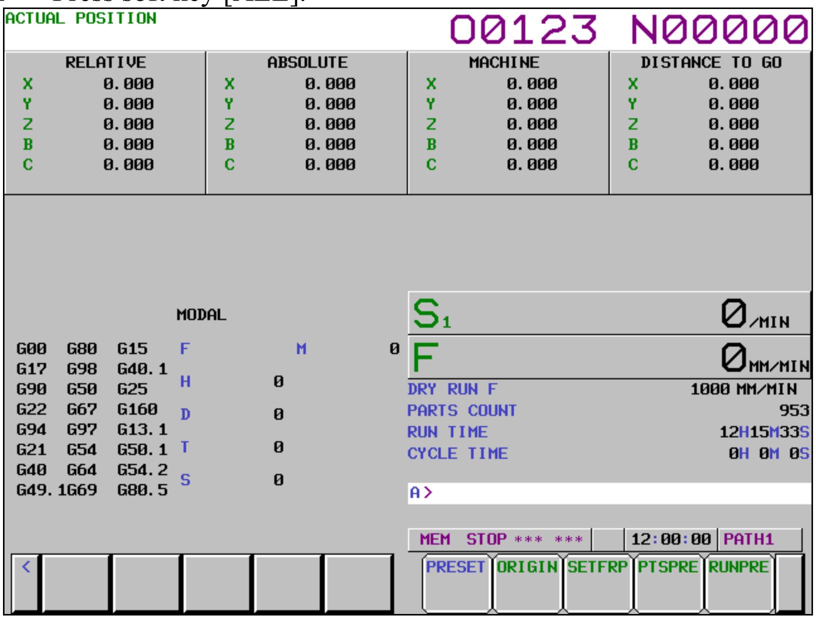


Fig. 12.1.3 (a) Current position (overall) screen (M series)

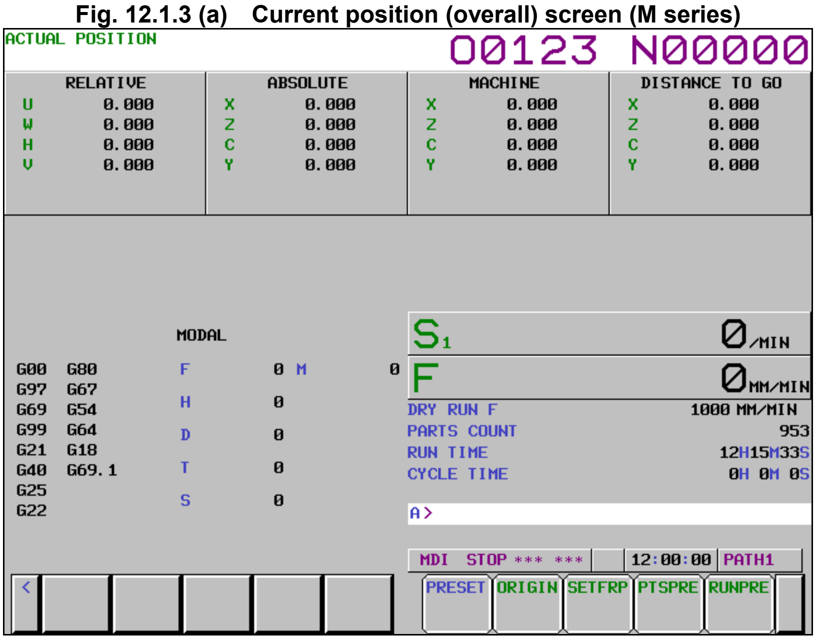


Fig. 12.1.3 (b) Current position (overall) screen (T series)

Explanation

- Coordinate display

The current positions of the tool in the following coordinate systems are displayed at the same time:

- Current position in the relative coordinate system (relative coordinate)
- Current position in the work coordinate system (absolute coordinate)
- Current position in the machine coordinate system (machine coordinate)
- Distance to go (distance to go)

- Distance to go

The distance remaining is displayed in the MEMORY or MDI mode. The distance the tool is yet to be moved in the current block is displayed.

- Machine coordinate system

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting parameter MCN (No. 3104#0).

- Resetting the relative coordinates

The total position display screen also supports the resetting of the relative coordinates to 0 or presetting of them to specified values. See the procedure for resetting the relative coordinates described in Subsection III-12.1.2

12.1.4 Workpiece Coordinate System Preset

A workpiece coordinate system shifted by an operation such as manual intervention can be preset using MDI operations to a pre-shift workpiece coordinate system. The latter coordinate system is displaced from the machine zero point by a workpiece origin offset value.
A command (G92.1) can be programmed to preset a workpiece coordinate system. (See II-7.2.4 in the section for programming.)

Procedure for the workpiece coordinate system preset

Procedure

1 Press function key POS .
2 Press soft key [(OPRT)].
3 Enter the axis name (X , Y , ...) and O .
4 Press soft key [PRESET].

Explanation

- Operation mode

This function can be executed when the reset state or automatic operation stop state is entered, regardless of the operation mode.

- Presetting relative coordinates

As with absolute coordinates, parameter PPD (No. 3104#3) is used to specify whether to preset relative coordinates (RELATIVE).

12.1.5 Actual Feedrate Display

The actual feedrate on the machine (per minute) can be displayed on a current position display screen or program check screen. On the 12 soft keys display unit, the actual feedrate is always displayed.

Display procedure for the actual feedrate on the current position display screen

Procedure

1 Press the function key POS to display a current position display screen. At the location indicated by [ ], an actual feedrate is displayed.

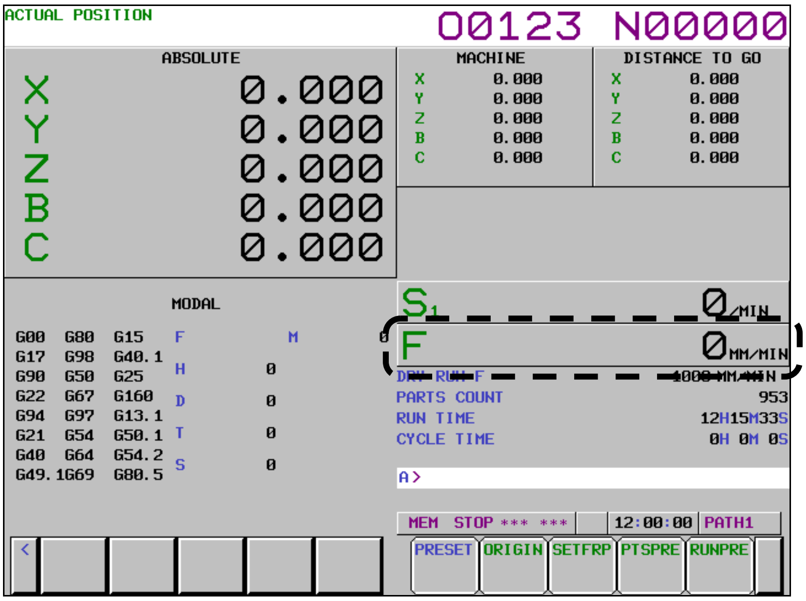


Fig. 12.1.5 (c) Current position (absolute) screen (M series)

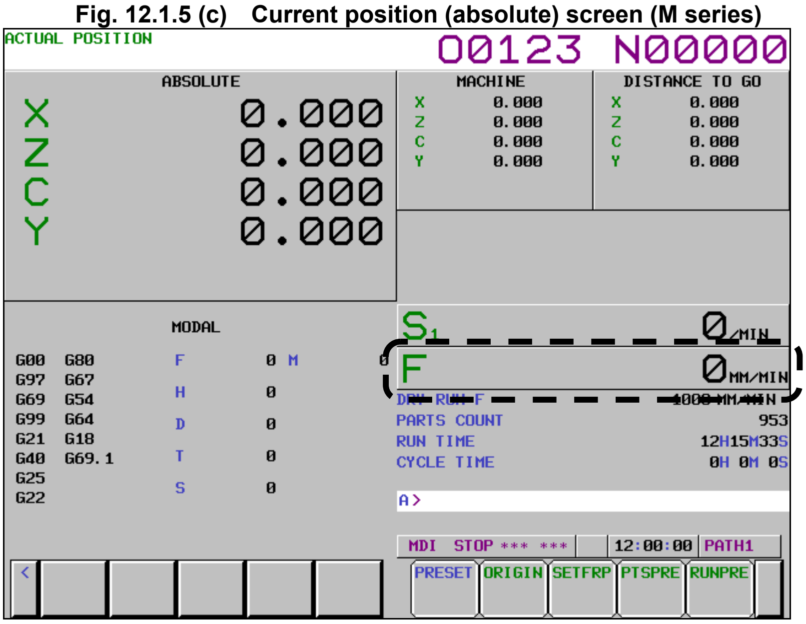


Fig. 12.1.5 (d) Current position (absolute) screen (T series)

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

Explanation

- Actual feedrate value

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n} (fi)^2}$$

n : Number of axes
fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate
Fact : Actual feedrate displayed

The display unit:

mm/min (metric input).
inch/min (Inch input, two digits below the decimal point are displayed.)

- Actual feedrate display of feed per revolution

In the case of feed per revolution and thread cutting, the actual feedrate displayed is the feed per minute rather than feed per revolution.

- Actual feedrate display of rotary axis

In the case of movement of rotary axis, the speed is displayed in units of deg/min but is displayed on the screen in units of input system at that time. For example, when the rotary axis moves at 50 deg/min, the following is displayed: 50 mm/min (in metric input) or 0.50 inch/min (in inch input).

- Actual feedrate display on the other screen

The program check screen also displays the actual feedrate.

12.1.6 Display of Run Time and Parts Count

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

Procedure

1 Press the function key POS to display a current position display screen. At the location indicated by , a run time and parts count are displayed.

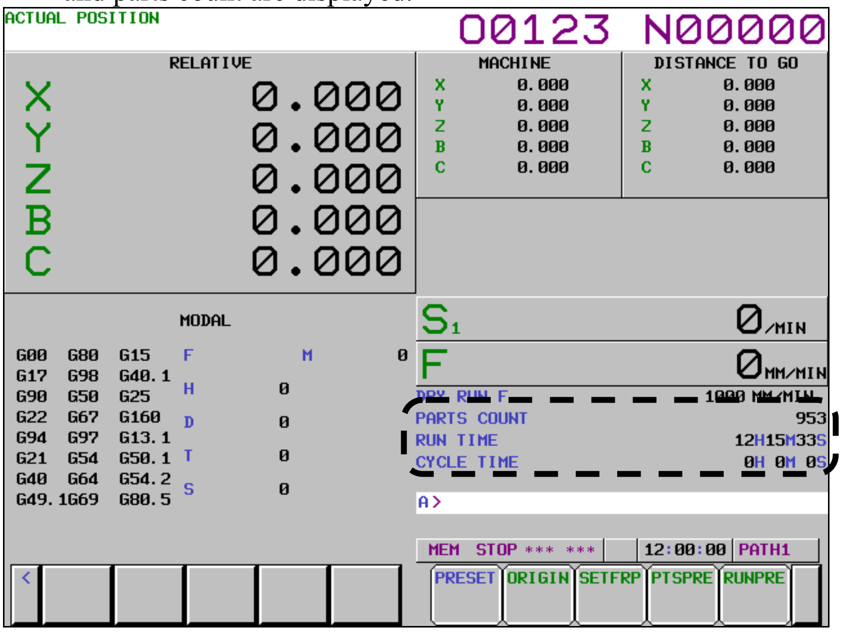


Fig. 12.1.6 (e) Current position (relative) screen (M series)

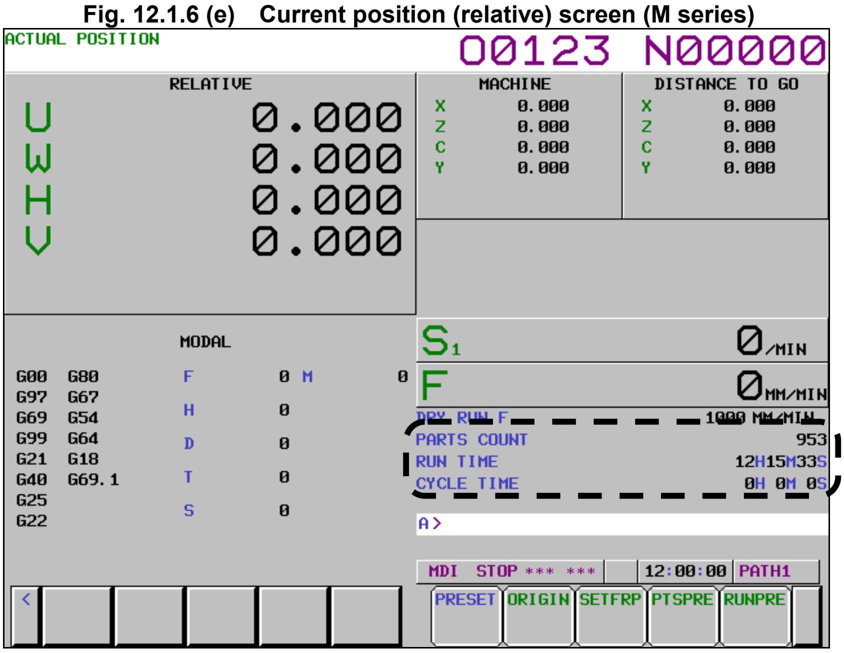


Fig. 12.1.6 (f) Current position (relative) screen (T series)

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanation

- PART COUNT

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- Display on the other screen

Details of the run time and the number of machined parts are displayed on the setting screen. See III-12.3.3.

- Parameter setting

The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters No. 6711, 6751, and 6752 or on the setting screen.

- Incrementing the number of machined parts

Parameter PCM (No. 6700#0) is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

12.1.7 Setting the Floating Reference Position

To perform floating reference position return with a G30.1 command, the floating reference position must be set beforehand.

Procedure for setting the floating reference position

Procedure

1. Press function key POS to display a screen used for displaying the current position.
2. Move the tool to the floating reference position by jogging.
3. Press soft key [(OPRT)].
4. Press soft key [SET FRP].
5. To register the floating reference positions for all axes, press soft key [ALLEXE].
   To register the floating reference position of a specific axis, enter the name of the axis (X , Y , etc.), then press soft key [EXEC]. Two or more names can be entered consecutively (e.g., X Y Z [EXEC]).
   The above operation stores the floating reference position. It can be checked with parameter (No. 1244).
6. In step 4, the floating reference position along a specified axis can also be stored by entering the axis name (such as X or Y) and pressing soft key [SET FRP].

Explanation

- Presetting the relative coordinate system

By parameter FPC (No. 1201#3), the relative position can be preset to 0 when a floating reference position is registered.

12.1.8 Operating Monitor Display

The reading on the load meter can be displayed for each servo axis and the serial spindle by setting parameter OPM (No. 3111#5) to 1. The reading on the speedometer can also be displayed for the serial spindle.

Procedure for displaying the operating monitor

Procedure

1 Press function key POS .

2 Press the continuous menu key .

3 Press soft key [MONI].

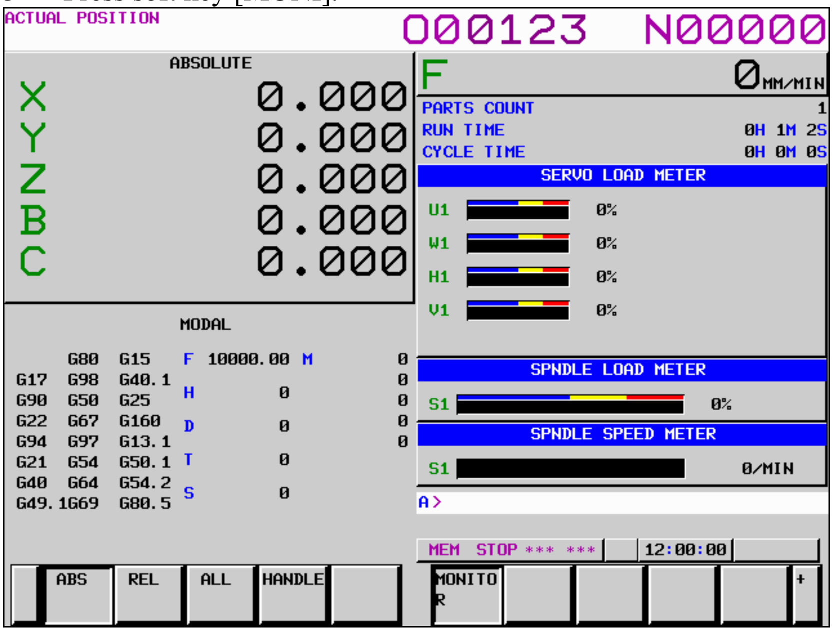


Fig. 12.1.8 (g) Operating monitor (M series)

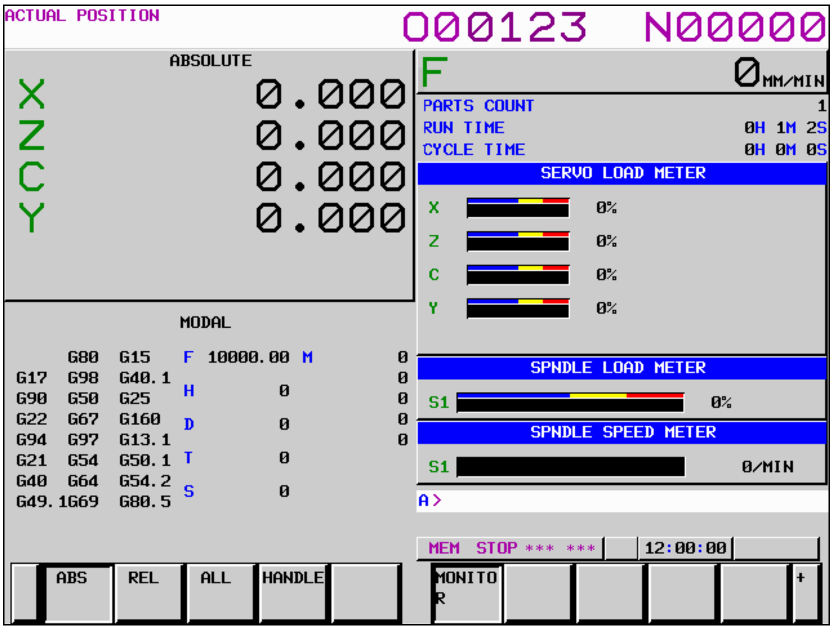


Fig. 12.1.8 (h) Operating monitor (T series)

Explanation

- Display of the servo axes

Servo axis load meters as many as the maximum number of controlled axes of the path can be displayed. One screen displays load meters for up to five axes at a time. By pressing the [MONITOR] soft key, load meters for the 6th axis and up are displayed.

- Display of the spindle axes

When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- Unit of graph

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%). The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- Load meter

The reading on the load meter depends on servo parameter No. 2086 and spindle parameter No. 4127.

- Speedometer

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting parameter OPS (No. 3111#6) to 1.
The spindle speed to be displayed during operation monitoring is calculated from the speed of the spindle motor (see the formula below). The spindle speed can therefore be displayed, during operation monitoring, even when no position coder is used. To display the correct spindle speed, however, the maximum spindle speed for each gear (spindle speed at each gear ratio when the spindle motor rotates at the maximum speed) must be set in parameters No. 3741 to No.3744.
The input of the clutch and gear signals for the first serial spindle is used to determine the gear which is currently selected. Control the input of the CTH1A and CTH2A signals according to the gear selection, by referring to the table below.
(Formula for calculating the spindle speed to be displayed)

$$\text{Spindle speed displayed during operation monitoring} = \frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor}} \times \text{Maximum spindle speed with the gear being used}$$

The following table lists the correspondence between clutch and gear selection signals CTH1A and CTH2A, used to determine the gear being used, and parameters:

CTH1A	CTH2A	Parameter	Serial spindle specification
0	0	=No.3741 (Maximum spindle speed with gear 1)	HIGH
0	1	=No.3742 (Maximum spindle speed with gear 2)	MEDIUM HIGH
1	0	=No.3743 (Maximum spindle speed with gear 3)	MEDIUM LOW
1	1	=No.3744 (Maximum spindle speed with gear 4)	LOW

The speed of the spindle motor and spindle can be displayed, during operation monitoring, only for the first serial spindle and the spindle switching axis for the first serial spindle. It cannot be displayed for the second spindle.

- Color of graph

If the value of a load meter exceeds 100%, the bar graph turns purple.

12.1.9 Display of Manual Feed for 5-axis Machining (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount)

The absolute coordinates of the tool tip, the number of pulses, and a machine axis move amount based on manual feed for 5-axis machining are displayed.

Displaying the screen for manual feed for 5-axis machining

Procedure

1 Press function key POS .

2 Press the continuous menu key several times to display the [5AXIS MANUAL] soft key.

3 Press the [5AXIS MANUAL] soft key to display the screen of manual feed for 5-axis machining.

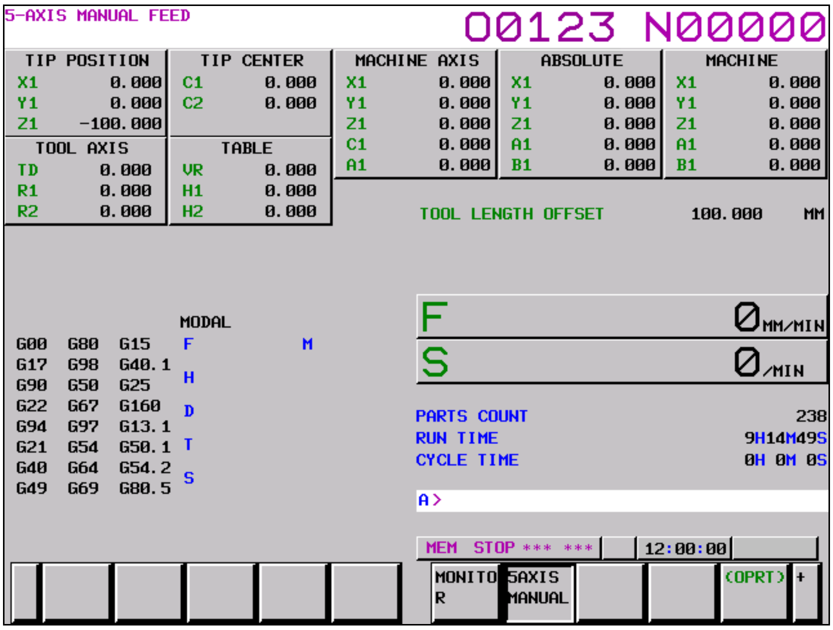


Fig. 12.1.9 (a) Screen of manual feed for 5-axis machining

Explanation

- Tool tip position

The addresses of the three basic machine configuration axes for performing manual feed for 5-axis machining and the current position of the tool tip are displayed.

- Tool axis reference (number of pulses)

TD

The amount of travel in the tool axis direction in tool axis direction handle feed, tool axis direction jog feed, or tool axis direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

R1

The amount of travel in the first axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis right-angle direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

R2

The amount of travel in the second axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis normal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Tool tip center (number of pulses)

C1

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the first rotary axis is displayed.
The unit is the least input increment of the first rotary axis.

C2

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the second rotary axis is displayed. The unit is the least input increment of the second rotary axis.

- Table reference (number of pulses)

VR

The amount of travel in the table reference vertical direction in table reference vertical direction handle feed, table reference vertical direction jog feed, or table reference vertical direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

H1

The amount of travel in the first axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

H2

The amount of travel in the second axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Amounts of machine axis travel

The addresses of machine configuration axes used for manual feed for 5-axis machining and the sum of the amount of travel of each axis used for manual feed for 5-axis machining are displayed.
The values of the basic three axes (X-axis, Y-axis, and Z-axis), the first rotary axis, and the second rotary axis are displayed in this order.
For the definition of the first rotary axis and second rotary axis, see the description of parameter No. 19680.
When bit 0 (CLR) of parameter No. 13113 is set to 1, the displayed data is cleared by a reset.

- Absolute coordinates, machine coordinates

The absolute coordinates and machine coordinates of all axes are displayed. If too many axes are involved for display on one screen, the remaining axes can be displayed by pressing the [5AXIS MANUAL] soft key for page feed.

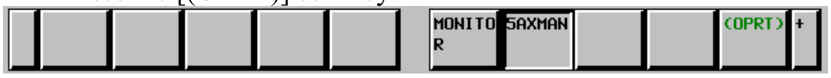
- F (feedrate)

- When bit 3 (CFD) of parameter No. 13113 is set to 0
  The composite feedrate at a control point on a linear axis or rotary axis is displayed.
- When bit 3 (CFD) of parameter No. 13113 is set to 1
  The feedrate of the tool tip is displayed.

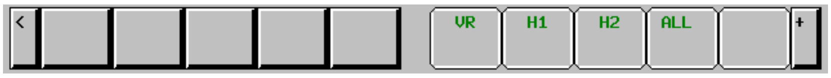
Operation

The display of the number of pulses can be cleared by soft key operations.

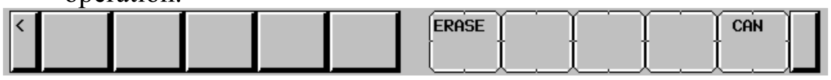
1 Press the [(OPRT)] soft key.



2 Select the soft key corresponding to a function subject to clearing of the amount of travel. Pressing the rightmost soft key displays the second page.



3 Press the [ERASE] soft key to clear the amount of travel of the specified function. Press the [CAN] soft key to cancel erase operation.



12.2 SCREENS DISPLAYED BY FUNCTION KEY PROG

This section describes the screens displayed by pressing function key PROG . The screens include a program editing screen, program folder list display screen, and screens for displaying the command states of the program currently being executed.

1. Program screen
2. Program list screen
3. Next block display screen
4. Program check screen

On the program screen, you edit the program that is currently selected, and display the block that is currently executed during program operation. In MDI mode, you also edit an MDI operation program, and display the block that is currently executed.

12.2.1 Program Contents Display

Displays the program currently being executed in MEMORY mode.

Displaying the program being executed

Procedure

1 Press function key PROG to display the program screen.

2 Press chapter selection soft key [PROGRAM].
The cursor is positioned at the block currently being executed.

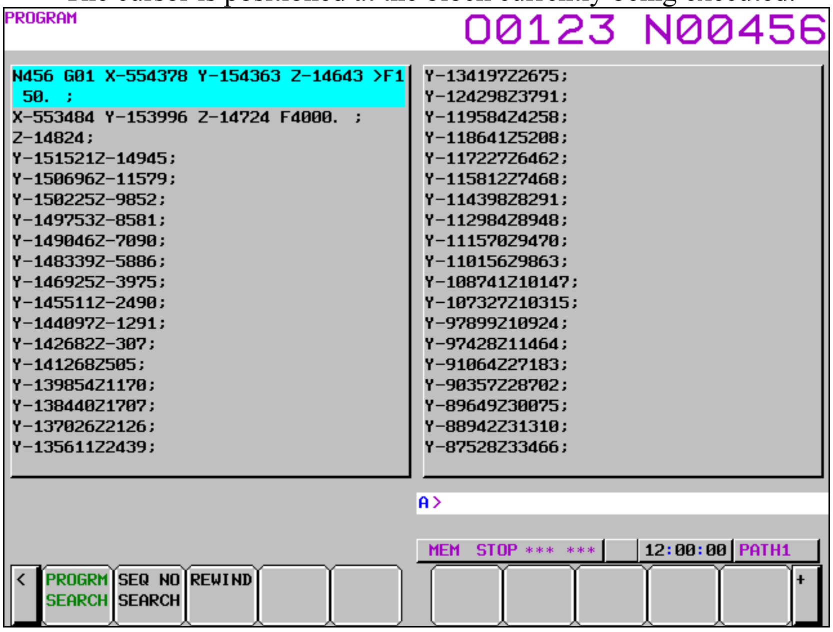


Fig. 12.2.1 (a) Screen for displaying the program being executed

12.2.2 Editing a Program

A program can be edited in the EDIT mode.
Two modes of editing are available. One mode is word editing, which performs word-by-word editing. The other is character editing, which performs character-by-character editing.
For program creation and editing operation, see Chapter III-9, "CREATING PROGRAMS " and Chapter III-10, "EDITING PROGRAMS".

Displaying the program editing screen

Procedure

1 Press function key PROG to display the program screen.

2 Press chapter selection soft key [PROGRAM].

- Word editing

Editing operations such as text insertion, modification, and deletion, and cursor movements are performed on a word-by-word basis.

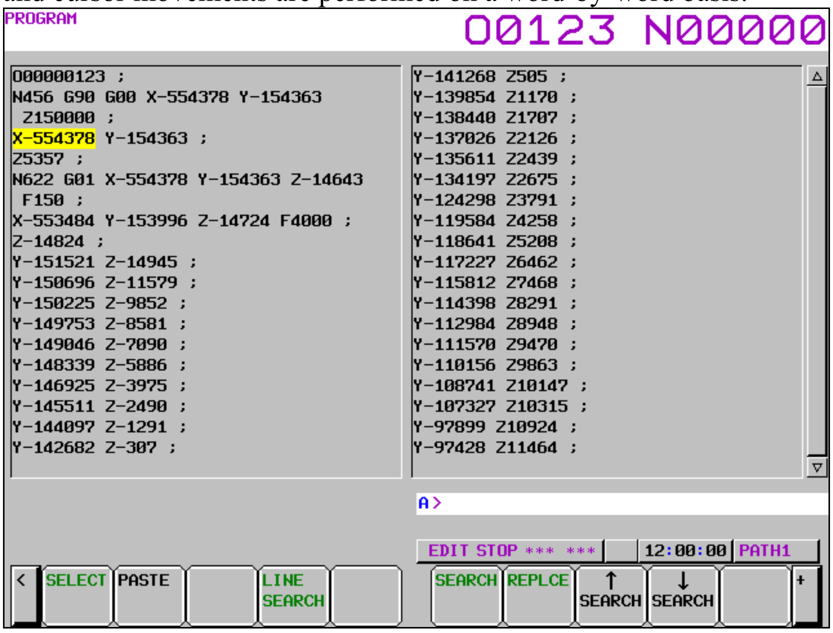


Fig. 12.2.2 (a) Program word editing screen

- Character editing

Program editing operations and cursor movements are performed on a character-by-character basis as with a general text editor.
Text is input directly to the cursor position instead of using the key input buffer.

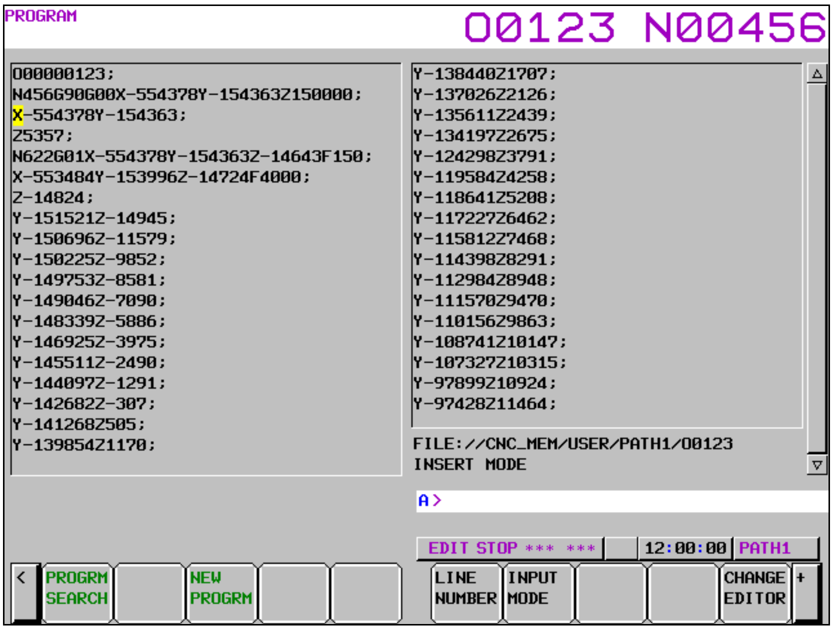


Fig. 12.2.2 (b) Program character editing screen

Switching between program editing modes

You can switch between word editing and character editing with soft keys.

Procedure

1 Press function key PROG to display the program screen.
2 Press chapter selection soft key [PROGRAM].
3 Press operation soft key [(OPRT)].
4 Pressing the [CHANGE EDITOR] operation soft key switches the editing mode between word editing and character editing.

12.2.3 Program Screen for MDI Operation

During MDI operation or editing of an MDI operation program in the MDI mode, the program currently being executed mode is displayed. For MDI operation, see Section III-4.2, "MDI Operation".

Procedure for displaying the program screen for MDI operation

Procedure

1 Press function key PROG .

2 Press chapter selection soft key [PROGRAM].
The program input from the MDI is displayed.

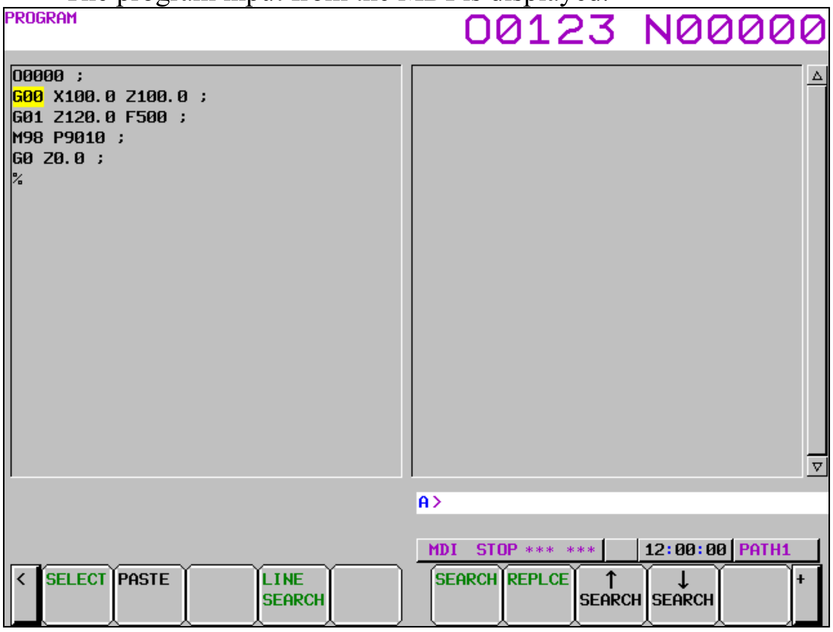


Fig. 12.2.3 (a) MDI operation program screen

12.2.4 Program Folder Screen

A list of programs registered in the program memory is displayed.
For the program folder screen, see Chapter III-11, "PROGRAM MANAGEMENT".

Displaying the program folder screen

Procedure

1 Press function key PROG .

2 Press chapter selection soft key [FOLDER].

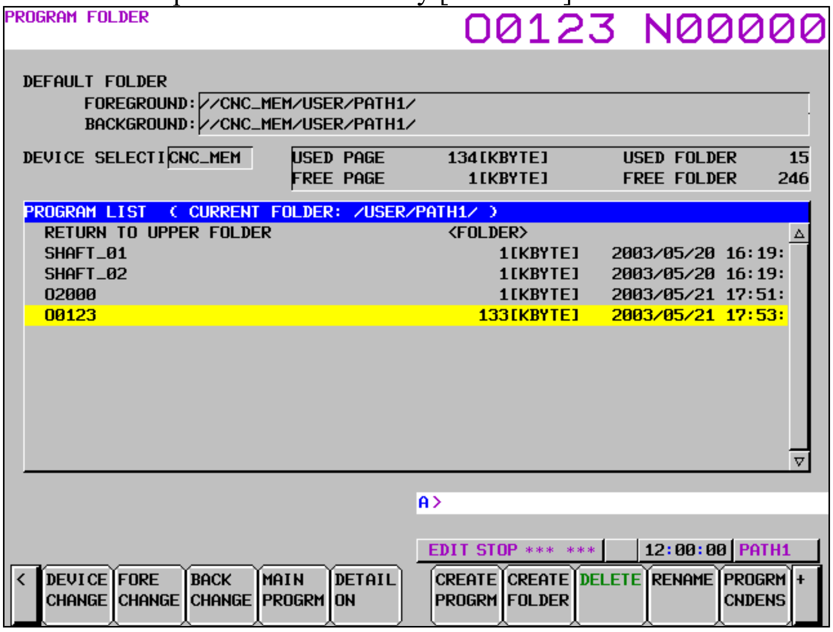


Fig. 12.2.4 (a) Program folder screen

12.2.5 Next Block Display Screen

Displays the block currently being executed and the block to be executed next.

Procedure for displaying the next block display screen

Procedure

1 Press function key PROG .

2 Press chapter selection soft key [NEXT].
The G codes, addresses, command values specified in the block currently being executed and the next block are displayed.

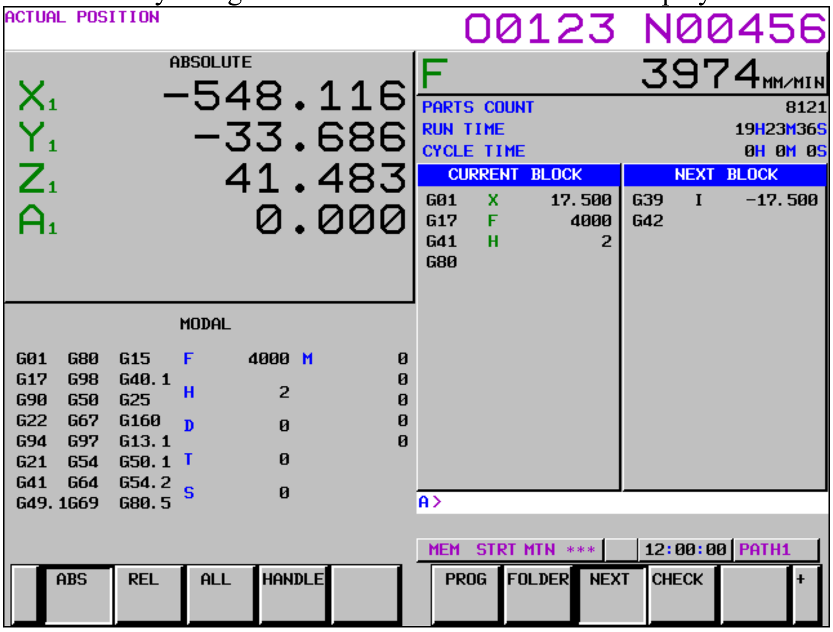


Fig. 12.2.5 (a) Next block display screen

12.2.6 Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data.

Procedure for displaying the program check screen

Procedure

1 Press function key PROG .

2 Press chapter selection soft key [CHECK].
The program currently being executed, current position of the tool, and modal data are displayed.

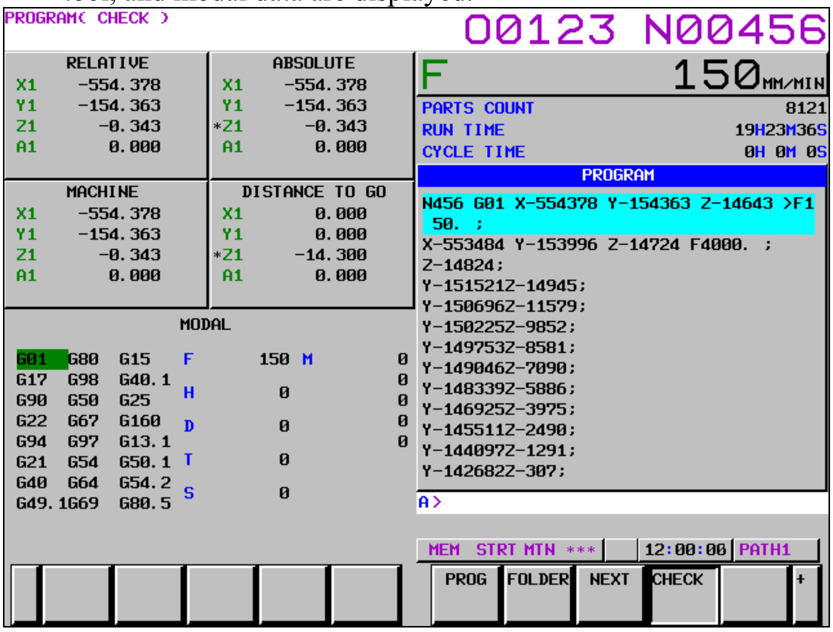


Fig. 12.2.6 (a) Program check screen

Explanation

- Program display

The program currently being executed is displayed.
The block being executed is displayed in reverse video.

- Current position display

The current position in the relative coordinate system, workpiece coordinate system, and machine coordinate system, and the remaining amount of travel are displayed.

- Modal G codes

Up to 24 modal G codes are displayed.

12.2.7 Background Editing

Editing one program during execution of another program is referred to as background editing. You can perform the same edit operations in the background as those in normal editing (foreground editing).

On a 10.4” or 15” display, you can perform background editing for the programs displayed in multiple tiled windows. You can switch from a program displayed in one of the tiled windows to another to perform copy, paste, and other edit operations, which allows you to edit programs efficiently.
You can edit two programs simultaneously on a 10.4” display or four programs on a 15” display.

Function

- Background editing

A program other than the currently selected program can be edited.
Background editing can be performed in any mode.

- Program selected in the foreground

If the program selected in the foreground is specified as a program to be edited in the background, background editing is started in the read-only mode. The text at an arbitrary position of the program currently being executed can be checked.

- Switching from the program directory screen to the editing screen

By selecting a program from the program directory screen, background editing can be started.
On the program directory screen, move the cursor to a program to be edited in the background, then press the MDI key [INPUT]. The screen display switches to the background editing screen.
Background editing can be started without entering a program name.

- Background editing on a multi-path system

Background editing does not depend on the path.
Even if the path to be operated is changed using the path selection signal, the background editing currently being executed is just continued.

Display

When background editing starts, the ordinary program editing screen switches to the background editing screen.
When two or more programs are edited in the background, the screen is split to display these programs. For a 10.4” display, you can edit two programs simultaneously; for a 15” display, you can edit four programs simultaneously.

- Word editing

Fig. 12.2.7 (a) shows background word editing performed simultaneously for two programs (right and left programs). On the status line at the top of the window for each program, the program name and “BG-EDIT” (indicating background editing in progress) are displayed.
The status line of the program being edited is displayed in reverse video.

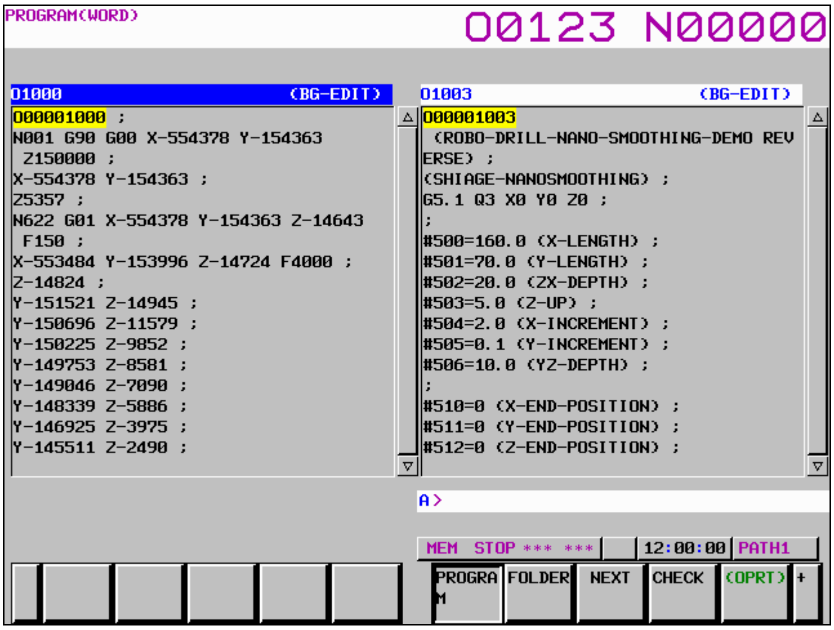


Fig. 12.2.7 (a) Background editing screen (word editing)

- Character editing

Fig.12.2.7(b) shows background character editing performed simultaneously for two programs (right and left programs). Similarly to word editing, at the top of the window for each program, the status line is displayed. In addition, the current input mode (INSERT MODE or OVERWRITE MODE) is displayed at the upper right of the screen for character editing.
The status line of the program being edited is displayed in reverse video.

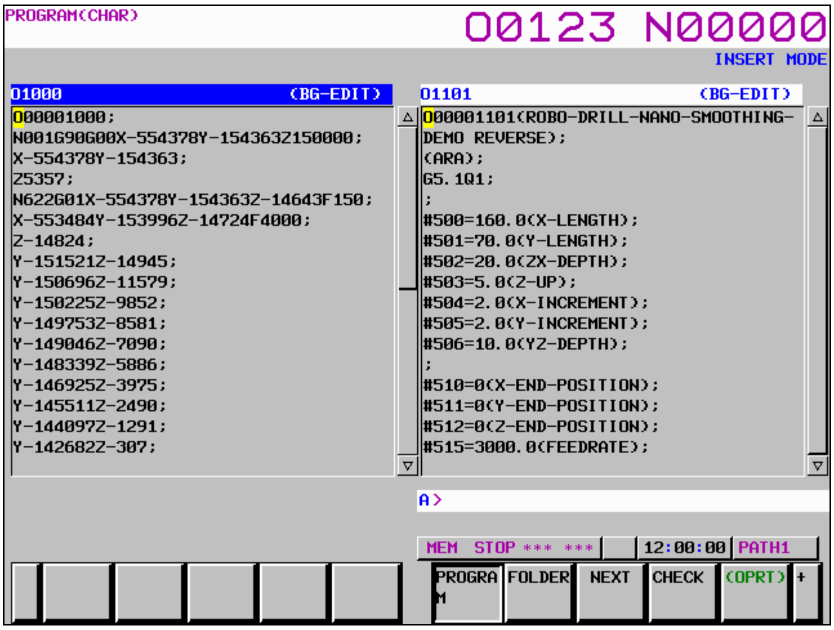


Fig. 12.2.7 (b) Background editing screen (character editing)

- Editing status

The following items are displayed on the status line and program editing area according to the background editing status.

Editing status	Displayed items
No program selected	(BG-EDIT) “NO PROGRAM” is displayed in the editing area.
Program opened	program-name + (BG-EDIT)
Read-only program opened	program-name + (BG:READ ONLY) The contents of the program are displayed in green.

Starting background editing from the editing screen

Procedure

Method 1

1 Press function key PROG .
2 Press soft key [PROG].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Press soft key [PROGRM SEARCH] to select a program to be edited.

Method 2 (only for word editing)

1 Press function key PROG .
2 Press soft key [PROG].
3 Key in the name of a program to be edited in the background.
4 Press soft key [BG EDIT].

When background editing has already started and editing start operation is performed again, the program being edited and newly specified program are edited simultaneously.

Explanation

When a program name is input in the key-in buffer, background editing for the program starts. When the specified program is not found, a new program is created and background editing starts.
When background editing is started with no program name specified, new background editing starts in the no program state. Perform a program search or create a new program.
When character editing is selected, background editing starts in the no program state first. Perform a program search or create a new program.

Starting background editing from the program directory screen

You can select a program from the program directory screen to start background editing. The cursor is used to select a program. You do not need to enter a program name.

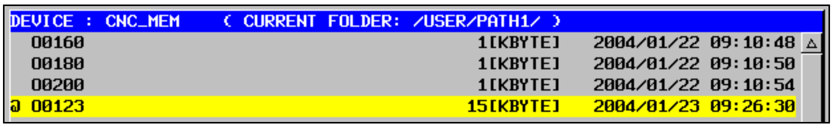
Procedure

1 Press function key PROG .

2 Press soft key [FOLDER].

3 Move the cursor to a program to be edited.

4 Press the MDI key [INPUT]. The screen display is switched to the background editing screen.



Background editing operation

- Editing operation

The same editing operations as performed in the foreground can be performed.

- Switching from a program to another for editing

To switch from a program to another for editing when editing multiple programs simultaneously in the background, press SHIFT, then ← or →.

- Editing operation in the read-only mode

Cursor movement and page switching are possible. However, program editing is disabled.

- Copying and pasting text between programs

Text in a program selected and copied during program editing is kept held after the program to be edited is changed. You can paste text in a program onto another program.

Background editing end operation

Background editing can be ended using the procedure described below.
The procedure for ending background editing of one program and that for ending all background editing of multiple programs are shown below.

- Ending editing of one program

1 Press function key PROG .
2 Press soft key [PROG].
3 Select a program for which to end editing.
4 Press soft key [(OPRT)], then soft key [BG EDIT].

- Ending editing of all programs

1 Press function key PROG .
2 Press soft key [PROG].
3 Press soft key [(OPRT)], then soft key [ALL BG END].

To return to ordinary foreground editing, end all background editing. If at least one program is being edited in the background, you cannot return to foreground editing.

12.2.8 Stamping the Machining Time

The execution times of the most recently executed ten programs can be displayed in hours, minutes, and seconds.
The calculated machining time can be inserted as a comment of the program to check the machining time on the program directory screen.

Procedure for stamping the machining time

Procedure

- Displaying the machining time

1 Press function key PROG .
2 Press the rightmost soft key ▷ several times to display soft key [TIME].
3 Press soft key [TIME]. The machining time display screen appears.

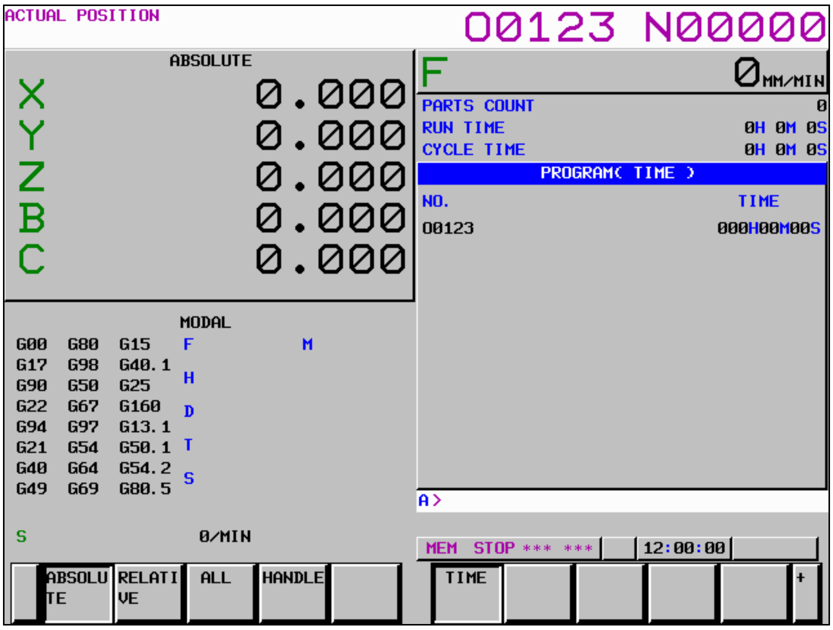


Fig. 12.2.8 (a) Machining time display screen

- Calculating the machining time

1 Select the memory operation mode, then press the RESET key.
2 Select the program screen, then select a program of which machining time you want to calculate.
3 Execute the program to perform actual machining.
4 When the RESET key is pressed or, M02 or M30 is executed, the machining time count operation stops. When the machining time display screen is selected, the program number of the stopped main program and its machining time are displayed.

5 The following figure shows the screen when the machining times of the ten main programs O0020, O0040, …, and O0200 are displayed and the screen when the machining time of O0220 is newly calculated after that.

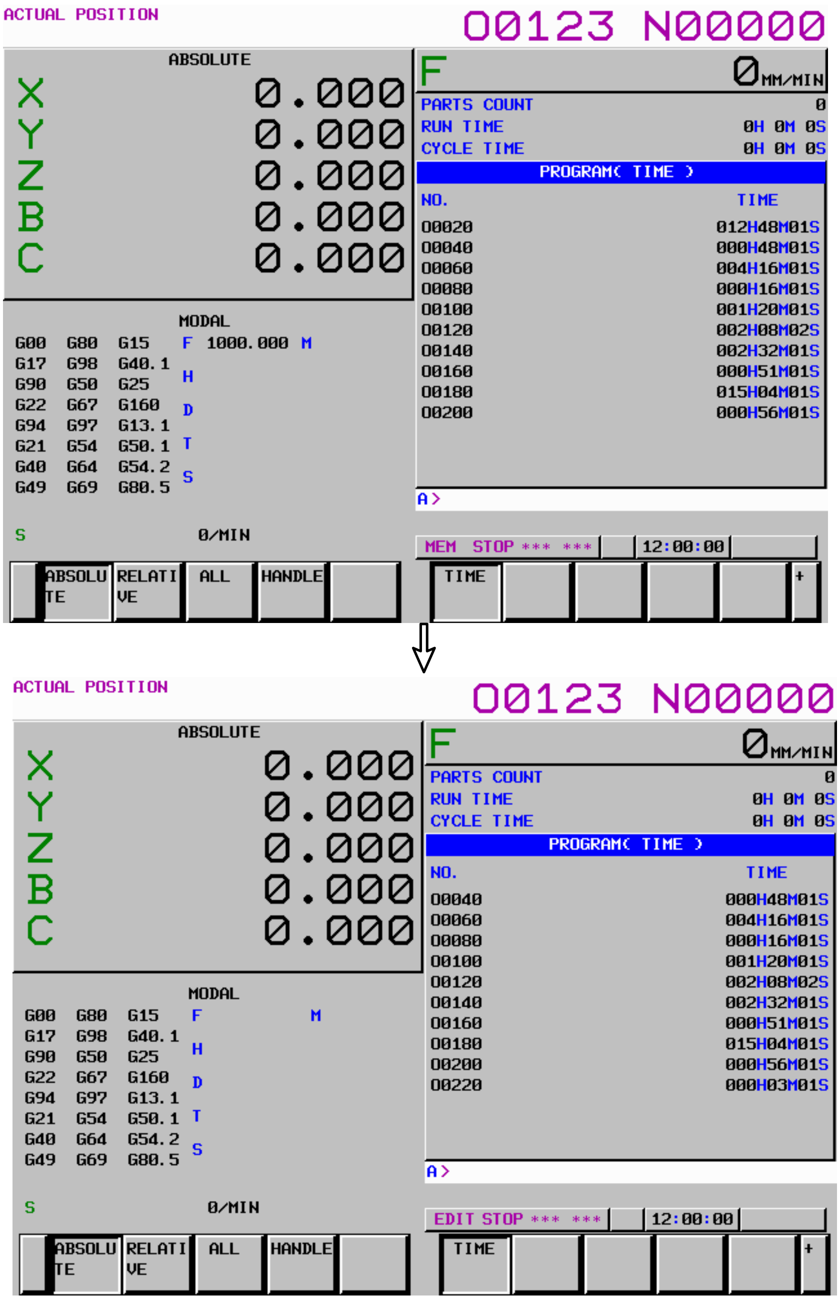


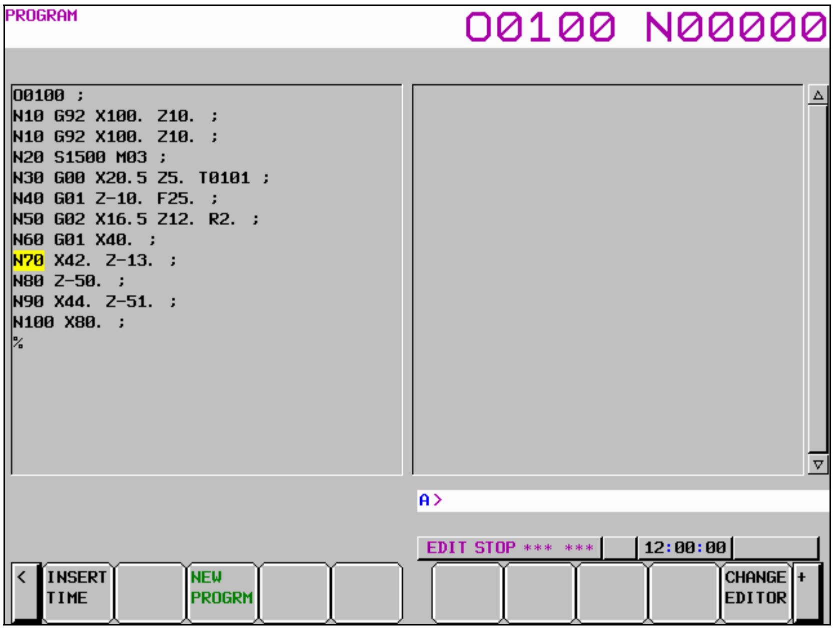
Fig. 12.2.8 (b) Stamping the machining time

Procedure for inserting the machining time on the program screen

Procedure

You can display the machining time of a program as a comment of the program. The procedure is shown below:

1 To insert the calculated machining time of a program as a comment, the machining time of the program must be displayed on the machining time display screen. Before stamping the machining time of the program, check that the program number is displayed on the machining time display screen.
2 Set the edit mode or background editing state and select the program screen. Then, select a program of which machining time you want to insert.
3 Suppose that the machining time of O0100 is displayed on the machining time display screen.
Press soft key [(OPRT)] to display the operation soft keys. Then, press the rightmost soft key several times to display soft key [INSERT TIME]. Press soft key [INSERT TIME]. The beginning of the program is displayed and the machining time of the program is inserted after the program name.



Press [INSERT TIME].

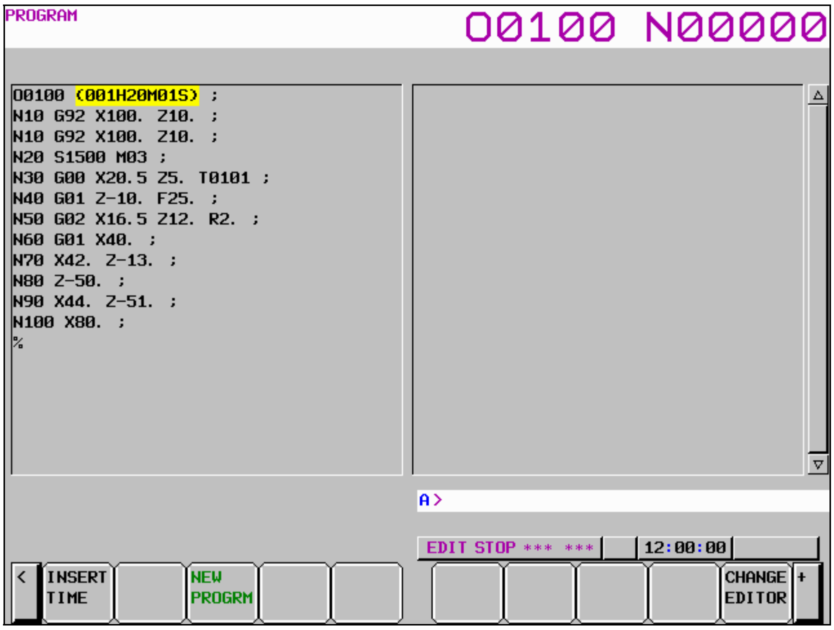
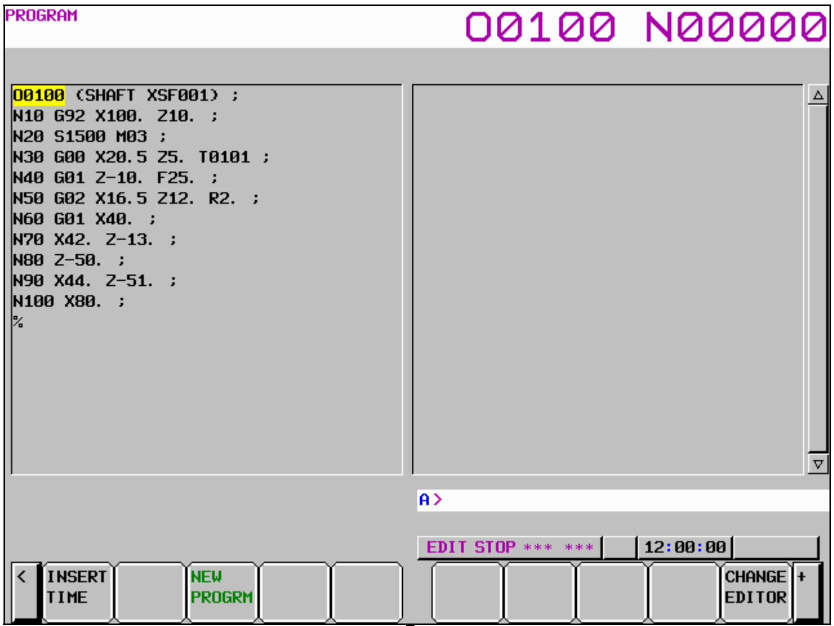


Fig. 12.2.8 (c)   Program screen

4 If a comment is written in the block containing the program number of a program of which machining time is to be inserted, the machining time is inserted after the comment.



Press [INSERT TIME].

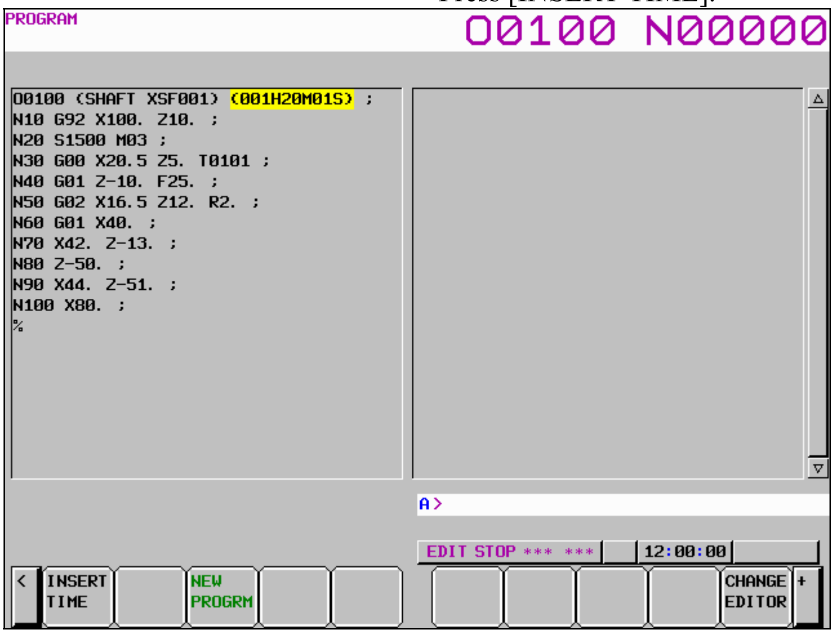


Fig. 12.2.8 (d) Program screen

Display on the program directory screen

The machining time of a program inserted in the program as a comment is displayed after the existing comment of the program on the program directory screen.

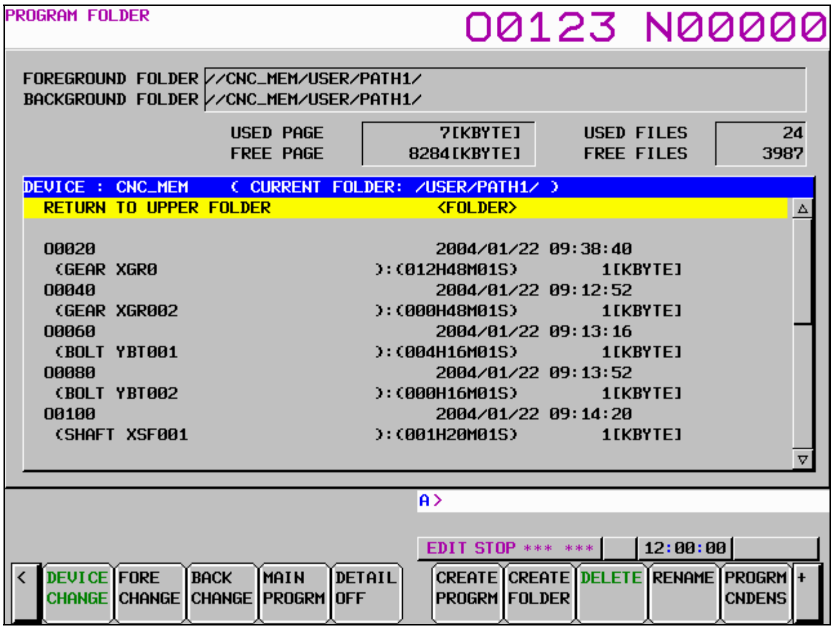


Fig. 12.2.8 (e) Program directory screen

Explanation

- Machining time

The machining time is counted from the initial start after a reset in the memory operation mode to the next reset. If a reset is not performed during operation, the machining time is counted from the start to M02 (or M30). The time during which operation is held is not counted, but the time used to wait for the completion of the M, S, T, and/or B functions is counted, however.

- Stamping the machining time

You can insert (stamp) the displayed machining time in a program stored in memory as a comment. The machining time is inserted as a comment after the program number.

- Program directory

You can display the machining time inserted after a program number on the program directory screen. This operation lets you know the machining time of each program to use the time as useful reference data when planning processes on site.

Limitation

- Alarm

The execution of a program may be held by an alarm during the machining time count operation. In this case, the machining time is counted until the alarm is released by a reset.

- M02

It may be specified that M02 does not reset the CNC, but returns completion signal FIN to the CNC to restart the program from the beginning successively (bit 5 (M02) of parameter No. 3404 is set to 0). In this case, when M02 returns completion signal FIN, the machining time count operation stops.

- Stamping the machining time

When an attempt is made to stamp the machining time of a program in the program, the machining time of the program may not be displayed on the machining time display screen. In this case, the machining time cannot be inserted in the program even if soft key [INSERT TIME] is pressed.

- Correcting the machining time

If an incorrect machining time is calculated (such as when a reset is made during the execution of a program), reexecute the program to calculate the correct machining time. The same program number may be displayed in two or more lines on the machining time display screen. In this case, the latest calculated machining time is inserted in the program.

- How a stamped machining time in each special state is displayed on the program directory screen

In the following states, the stamped machining time is displayed on the program directory screen as shown below.

1 When the comment of a program is longer than 16 characters
The 17th and subsequent characters of the comment are discarded and the machining time display field is left blank.

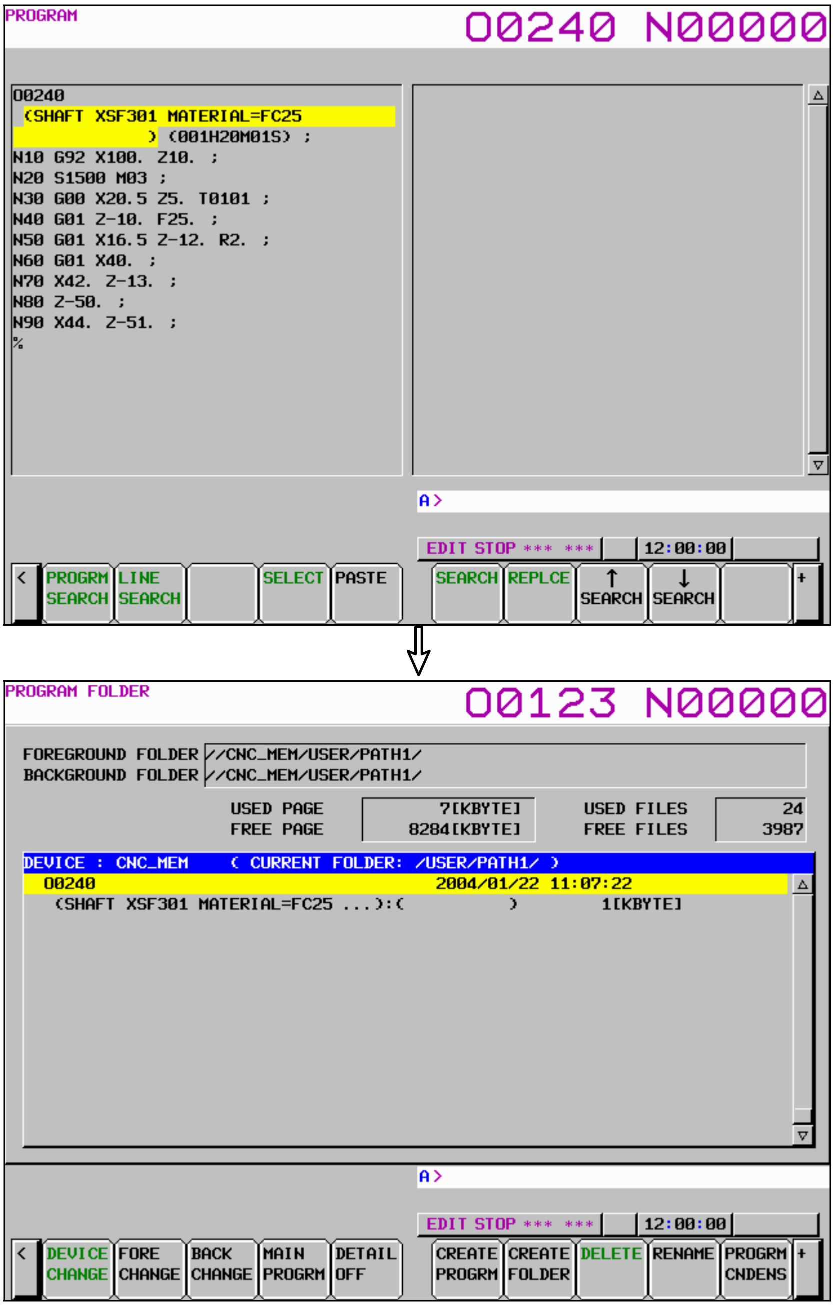


Fig. 12.2.8 (f) 1. When a program name is longer than 16 characters

2 When two or more machining times are stamped
The first machining time is displayed.

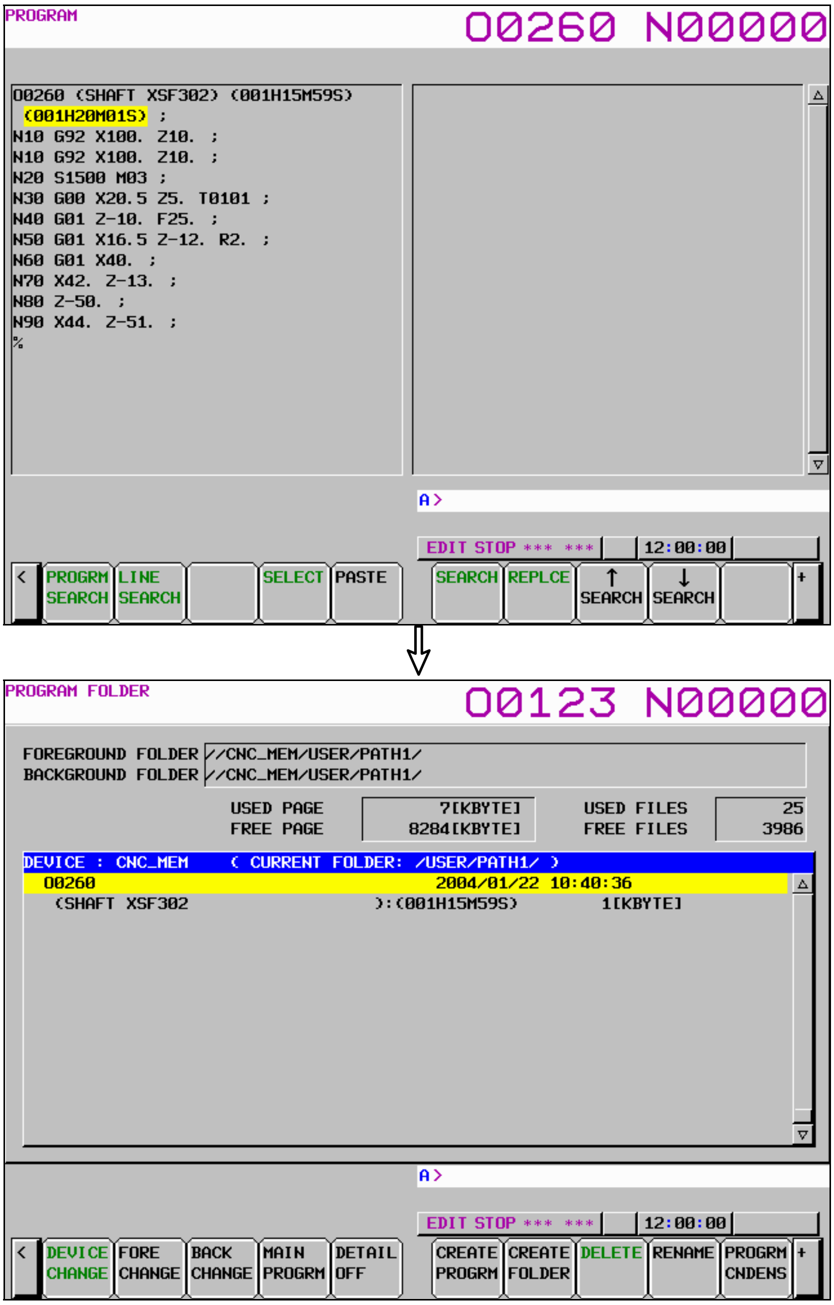


Fig. 12.2.8 (g) 2. When two or more machining times are stamped

3 When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order) The machining time display field is left blank.

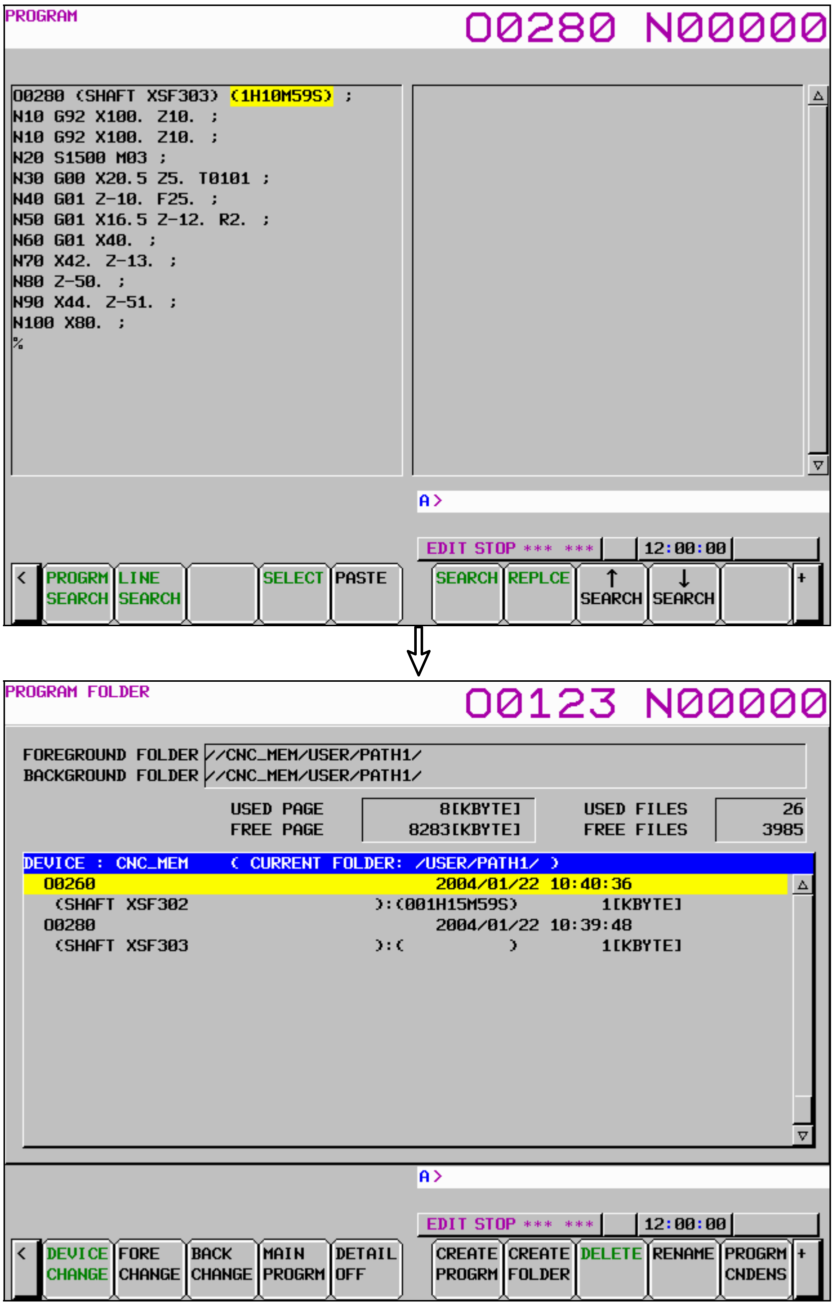


Fig. 12.2.8 (h) When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order)

12.3 SCREENS DISPLAYED BY FUNCTION KEY OFFSET SETTING

Press function key OFFSET SETTING to display or set tool compensation values and other data.
This section describes how to display or set the following data:

1. Tool compensation value
2. Settings
3. Sequence number comparison and stop
4. Run time and part count
4. Workpiece origin offset value
5. Custom macro common variables
7. Software operator's panel
8. Tool management data
9. Display language switching
10. Protection of data at eight levels
11. Precision level selection

The software operator's panel, display language switching, and precision level selection depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

12.3.1 Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.
See III-9.2 for automatic insertion of sequence numbers.
See III-12.3.2 for the sequence number comparison and stop function.
This subsection describes how to set data.

Procedure for setting the setting data

Procedure

1 Select the MDI mode.

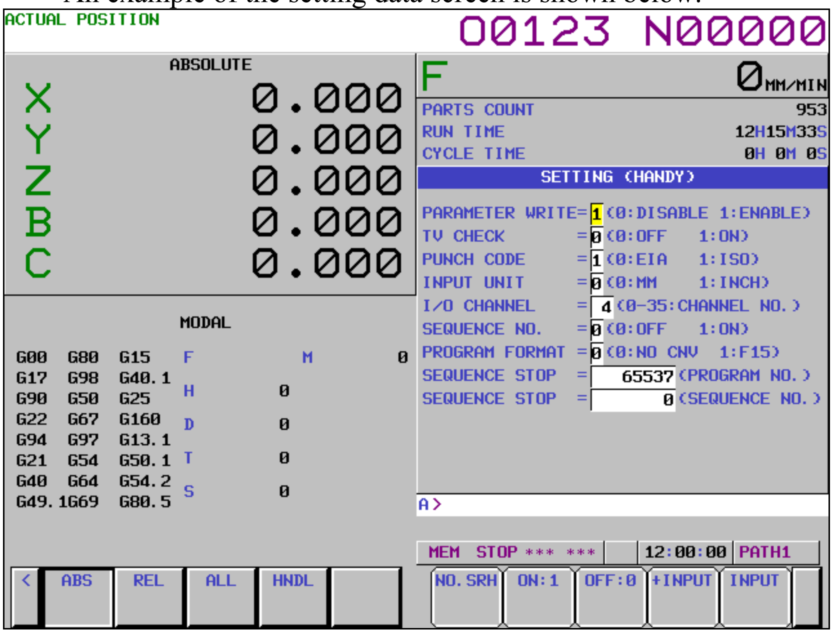
2 Press function key 

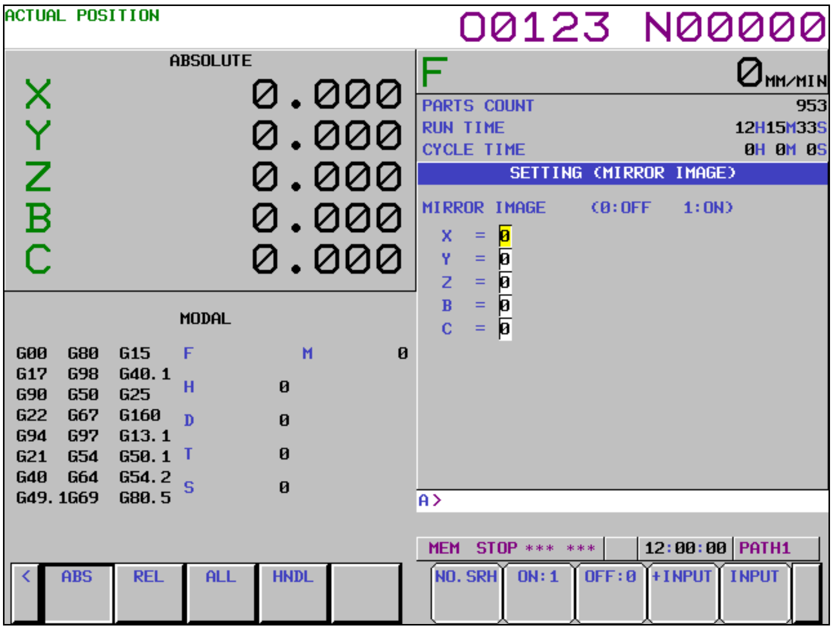
.

3 Press soft key [SETTING] to display the setting data screen.
This screen consists of several pages.
Press page key 

or 

until the desired screen is displayed.
An example of the setting data screen is shown below.





4 Move the cursor to the item to be changed by pressing cursor keys ↑ ↓ ← →.

5 Enter a new value and press soft key [INPUT].

Explanation

- PARAMETER WRITE

Setting whether parameter writing is enabled or disabled.
0 : Disabled
1 : Enabled

- TV CHECK

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- PUNCH CODE

Setting code when data is output through reader/puncher interface.
0 : EIA code output
1 : ISO code output

- INPUT UNIT

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- I/O CHANNEL

Using channel of reader/puncher interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2

- SEQUENCE NO.

Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
0 : Does not perform automatic sequence number insertion.
1 : Perform automatic sequence number insertion.

- PROGRAM FORMAT

Setting of whether to use the Series 15 format.
0: Uses the standard format.
1: Uses the Series 15 format.
For the Series 15 format, refer to Chapter II-6 in the User's Manual (T series) or Chapter II-7 of Part II in the User's Manual (M series).

- SEQUENCE STOP

Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs

- MIRROR IMAGE

Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on

- Others

Page key 

or 

can also be pressed to display the SETTING (TIMER) screen. See III-12.3.3 for this screen.

12.3.2 Sequence Number Comparison and Stop

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

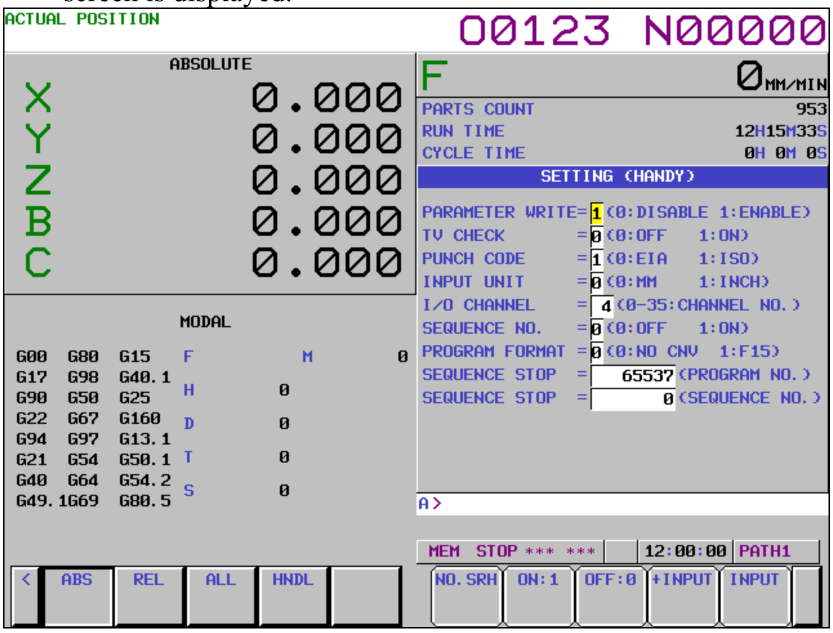
Procedure

1 Select the MDI mode.

2 Press function key OFFSET SETTING .

3 Press chapter selection soft key [SETTING].

4 Press page key PAGE ↑ or PAGE ↓ several times until the following screen is displayed.



5 Enter in (PROGRAM NO.) for SEQUENCE STOP the number (1 to 99999999) of the program containing the sequence number with which operation stops.

6 Enter in (SEQUENCE NO.) for SEQUENCE STOP (with eight or less digits) the sequence number with which operation is stopped.

7 When automatic operation is executed, operation enters single block mode at the block containing the sequence number which has been set.

Explanation

- Sequence number after the program is executed

After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one.

- Exceptional blocks

If the predetermined sequence number is found in a block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

[Example]

```
N1 #1=1 ;
N2 IF [#1 EQ 1] GOTO 08 ;
N3 GOTO 09 ;
N4 M98 P1000 ;
N5 M99 ;
```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.

- Stop in the canned cycle

If the predetermined sequence number is found in a block which has a canned cycle command, the execution of the program stops after the return operation is completed.

- When the same sequence number is found several times in the program

If the predetermined sequence number appears twice or more in a program, the execution of the program stops after the block in which the predetermined sequence number is found for the first time is executed.

- Block to be repeated a specified number of times

If the predetermined sequence number is found in a block which is to be executed repeatedly, the execution of the program stops after the block is executed specified times.

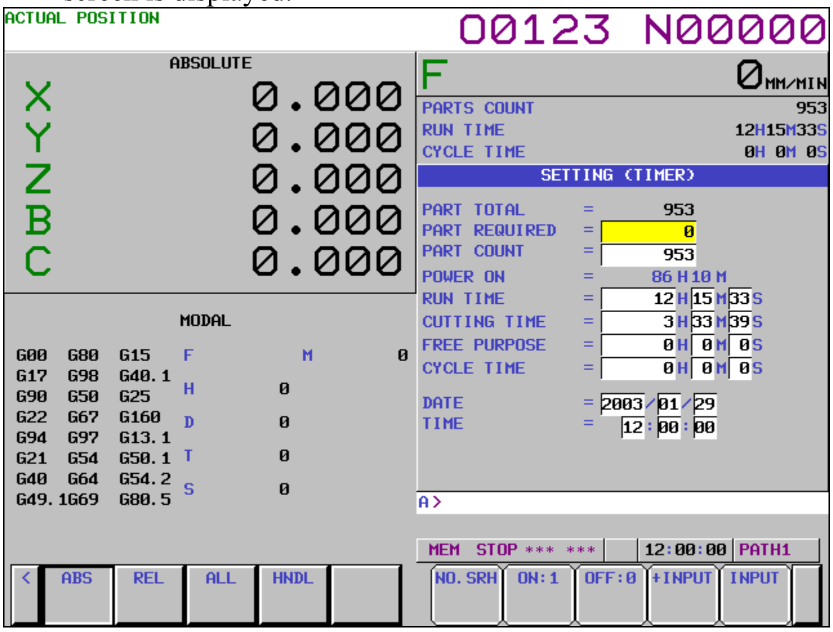
12.3.3 Displaying and Setting Run Time, Parts Count, and Time

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed. This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Procedure for Displaying and Setting Run Time, Parts Count and Time

Procedure

1 Select the MDI mode.
2 Press function key OFFSET SETTING .
3 Press chapter selection soft key [SETTING].
4 Press page key PAGE ↑ or PAGE ↓ several times until the following screen is displayed.



5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.
6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press soft key [INPUT].

Explanation

- PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. This value cannot be set on this screen. Set the value in parameter No. 6712.

- PARTS REQUIRED

It is used for setting the number of machined parts required.
When the "0" is set to it, there is no limitation to the number of parts.
Also, its setting can be made by the parameter No 6713.

- PARTS COUNT

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. The value can also be set by parameter No. 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- POWER ON

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter No. 6750.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter No. 6751 or No. 6752.

- CUTTING TIME

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter No. 6753 or No. 6754.

- FREE PURPOSE

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- DATA and TIME

Displays the current date and time. The date and time can be set on this screen.

- Usage

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter (No. 6710) is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (parameter PCM (No. 6700#0) is set to 1). For details, see the manual issued by machine tool builders.

Limitation

- Run time and part count settings

Negative value cannot be set. Also, the setting of "M" and "S" of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

- Time settings

Neither negative value nor the value exceeding the value in the following table can be set.

Table 12.3.3 (a)

Item	Maximum value	Item	Maximum value
Year	2096	Hour	23
Month	12	Minute	59
Day	31	Second	59

12.3.4 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59, G54.1 P1 to G54.1 P48 and G54.1 P1 to G54.1 P300) and external workpiece origin offset. The workpiece origin offset and external workpiece origin offset can be set on this screen.

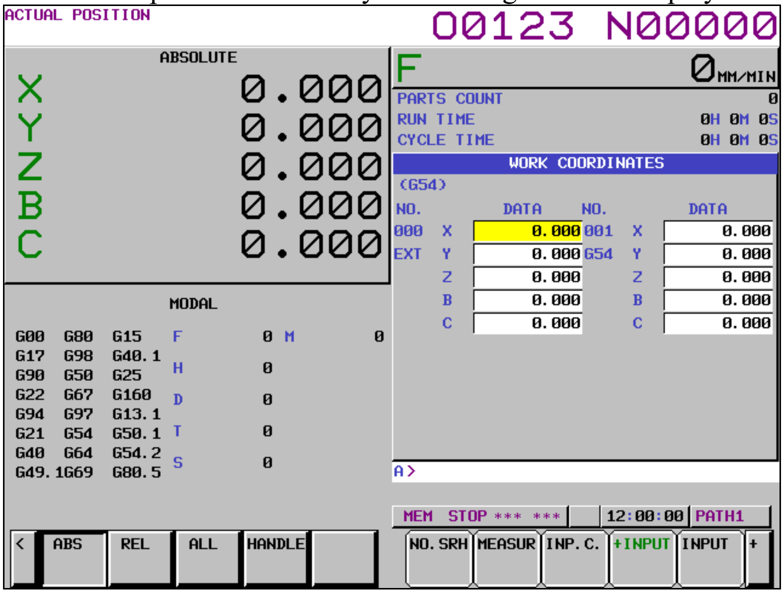
Procedure for displaying and setting the workpiece origin offset value

Procedure

1 Press function key 

.

2 Press chapter selection soft key [WORK].
The workpiece coordinate system setting screen is displayed.



3 The screen for displaying the workpiece origin offset values consists of two or more pages. Display a desired page in either of the following two ways:

- Press the page up 

or page down 

key.
- Enter the workpiece coordinate system number (0 : external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59, P1 to P48 : workpiece coordinate systems G54.1 P1 to G54.1 P48, P1 to P300 : workpiece coordinate systems G54.1 P1 to G54.1 P300) and press operation selection soft key [NO.SRH].

4 Turn off the data protection key to enable writing.

5 Move the cursor to the workpiece origin offset to be changed.

6 Enter a desired value by pressing numeric keys, then press soft key [INPUT]. The entered value is specified in the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing soft key [+INPUT], the entered value can be added to the previous offset value.

7 Repeat 5 and 6 to change other offset values.

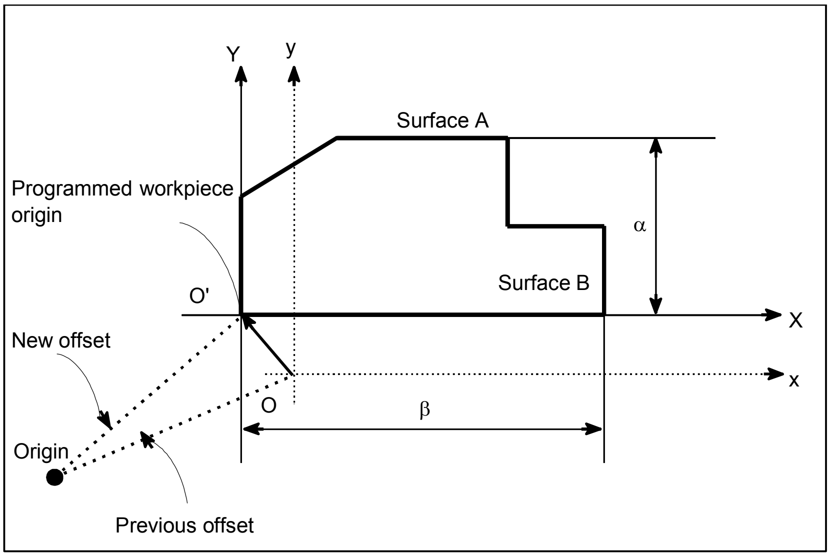
8 Turn on the data protection key to disable writing.

12.3.5 Direct Input of Workpiece Origin Offset value measured

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen such that the command values match the actual dimensions.
Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

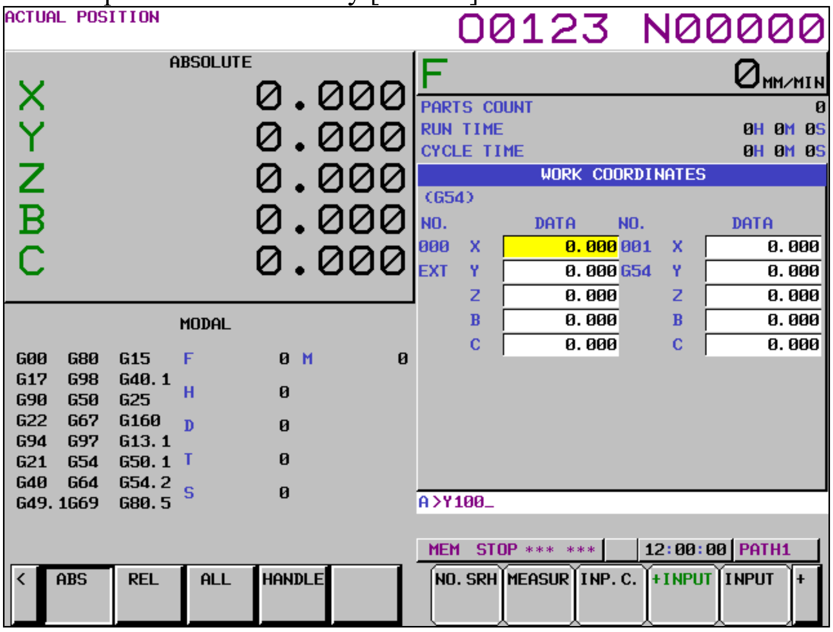
Procedure for direct input of workpiece origin offset value measured

Procedure



1 When the workpiece is shaped as shown above, position the reference tool manually until it touches surface A of the workpiece.
2 Retract the tool without changing the Y coordinate.
3 Measure distance α between surface A and the programmed origin of the workpiece coordinate system as shown above.
4 Press function key OFFSET SETTING .

5 To display the workpiece origin offset setting screen, press the chapter selection soft key [WORK].



6 Position the cursor to the workpiece origin offset value to be set.

7 Press the address key for the axis along which the offset is to be set (Y-axis in this example).

8 Enter the measured value (α) then press the [MEASUR] soft key.

9 Move the reference tool manually until it touches surface B of the workpiece.

10 Retract the tool without changing the X coordinate.

11 Enter the measured value of distance (β) to surface B as an X value as with steps 7 and 8.

Limitation

- Consecutive input

Offsets for two or more axes cannot be input at the same time.

- During program execution

This function cannot be used while a program is being executed.

12.3.6 Displaying and Setting Custom Macro Common Variables

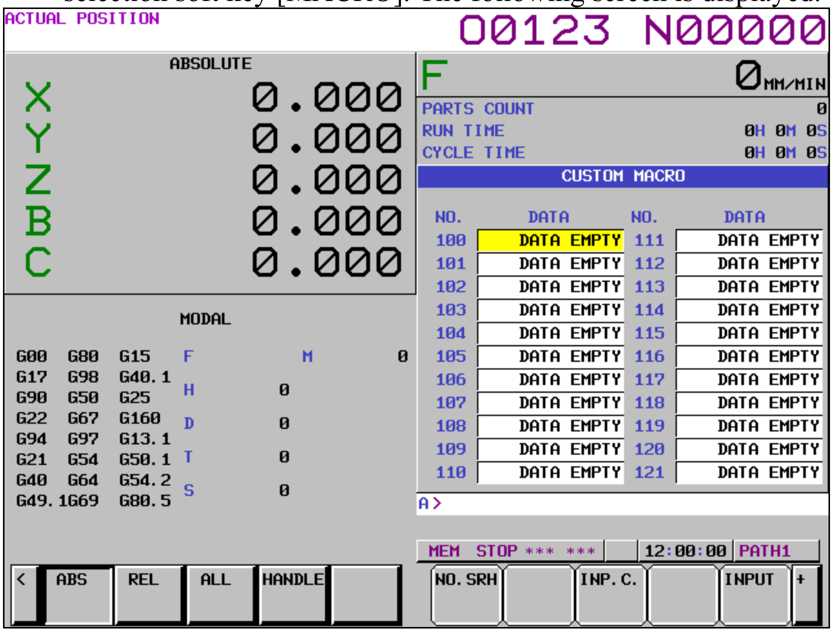
Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the screen. The values for variables can be set on this screen. Relative coordinates can also be set to variables.

Procedure for displaying and setting custom macro common variables

Procedure

1 Press function key OFFSET SETTING .

2 Press the continuous menu key ▷ , then press chapter selection soft key [MACRO]. The following screen is displayed.



3 Move the cursor to the variable number to set using either of the following methods:

- Enter the variable number and press soft key [NO.SRH].
- Move the cursor to the variable number to set by pressing page keys PAGE ↑ and/or PAGE ↓ and cursor keys ↑ , ↓ , ← , and/or → .

4 Enter data with numeric keys and press soft key [INPUT].

5 To set a relative coordinate in a variable, press address key X , Y , or Z , then press soft key [INP.C.].

6 To set a blank in a variable, just press soft key [INPUT]. The value field for the variable becomes blank.

Explanation

If the value of a variable produced by an operation is not displayable, an indication below is provided.

When the significant number of digits is 12 (with bit 0 (F16) of parameter No. 6008 set to 0):

Variable value range	Variable value indication
0 < Variable value < +0.00000000001	+Underflow
0 > Variable value > -0.00000000001	-Underflow
Variable value > 999999999999	+Overflow
Variable value < -999999999999	-Overflow

When the significant number of digits is 8 (with bit 0 (F16) of parameter No. 6008 set to 1):

Variable value range	Variable value indication
0 < Variable value < +0.0000001	+Underflow
0 > Variable value > -0.0000001	-Underflow
Variable value > 99999999	+Overflow
Variable value < -99999999	-Overflow

12.3.7 Displaying and Setting Real Time Custom Macro Data

Real time macro variables (RTM variables) are dedicated to real time custom macros.
RTM variables are divided into temporary real time macro variables (temporary RTM variables) and permanent real time macro variables (permanent RTM variables).
The values of permanent RTM variables are kept stored after the power is turned off.
Temporary RTM variables are cleared to 0 when the power is turned off.

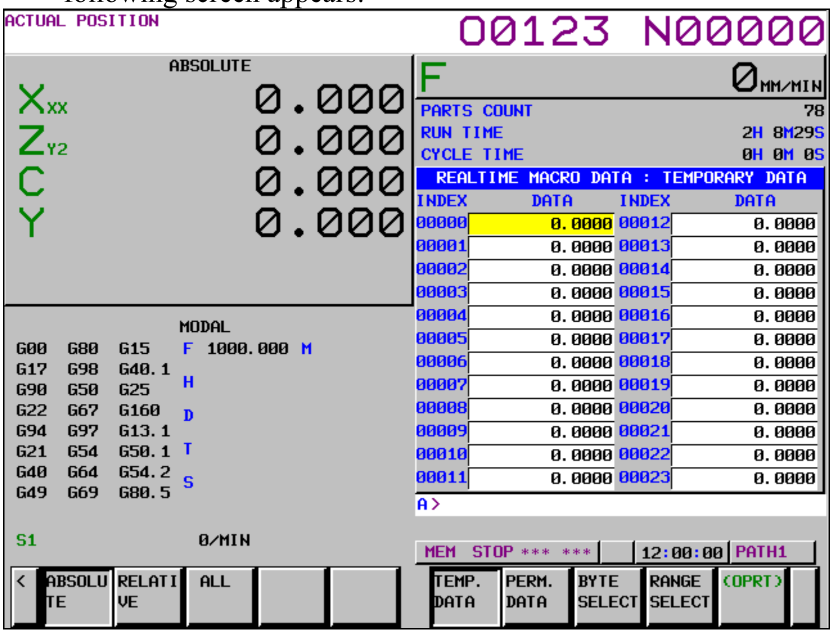
System variables (DI/DO variables) dedicated to real time custom macros are used to read and write PMC interface signals. Data can be read and written in bit and byte units. Before writing a signal, release the protection of the corresponding variable on the PMC signal protect screen.

Displaying and setting real time custom macro variables

Procedure

1 Press function key SYSTEM .

2 Press the continuous menu key several times, then press chapter selection soft key [REAL TIME MACRO]. The following screen appears:



3 To display or set real time custom macro variables of which values are not stored at power-off, press soft key [TEMP. DATA].

4 To display or set real time custom macro variables of which values are stored at power-off, press soft key [PERM. DATA].

5 Move the cursor to the number of a real time custom macro variable you want to set using either of the following methods:
- Enter the number of a real time custom macro variable and press soft key [NO. SRH].
- Move the cursor to the number of a real time custom macro variable you want to set by pressing page keys and/or and cursor keys , , , and/or .

6 Enter data.

Displaying and setting DI/DO variables

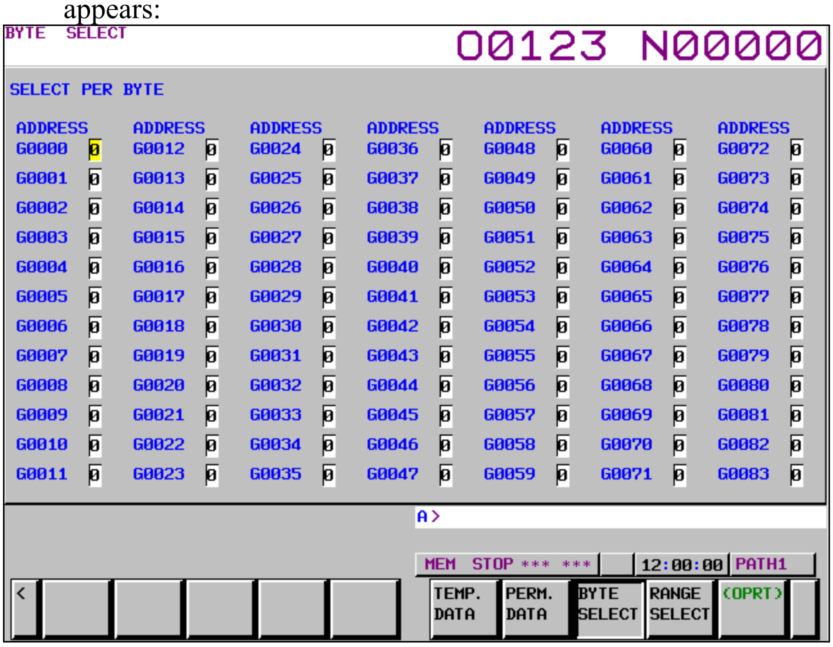
Procedure

For setting in byte units:

1 Press function key SYSTEM .

2 Press the continuous menu key several times, then press chapter selection soft key [REAL TIME MACRO]. The following screen appears:

3 Press soft key [BYTE SELECT]. The following screen appears:



4 Move the cursor to the number of a DI/DO variable you want to set using either of the following methods:
- Enter the number and press soft key [NO. SRH].
- Move the cursor to a desired number by pressing page keys and/or and cursor keys , , , and/or .

5 Enter data.

12.3.8 Displaying and Setting the Software Operator's Panel

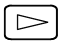
Operations on the MDI panel can substitute for the functions of switches on the machine operator's panel. This means that a mode selection, jog feed override selection, and so forth can be made with operations on the MDI panel, eliminating the need to use the corresponding switches on the machine operator's panel.
Jog feed can be performed using numeric keys.

Procedure for displaying and setting the software operator's panel

Procedure

1 Press function key 

.

2 Press the continuous menu key , then press chapter selection soft key [OPR].

3 The screen consists of several pages. Press page key 

or 

until the desired screen is displayed.

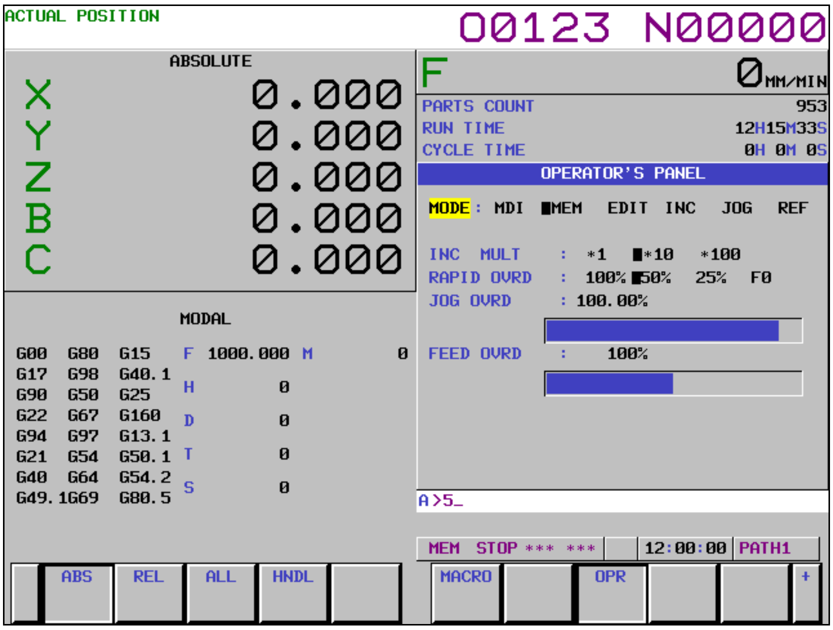


Fig. 12.3.8 (a) Without the manual handle feed function

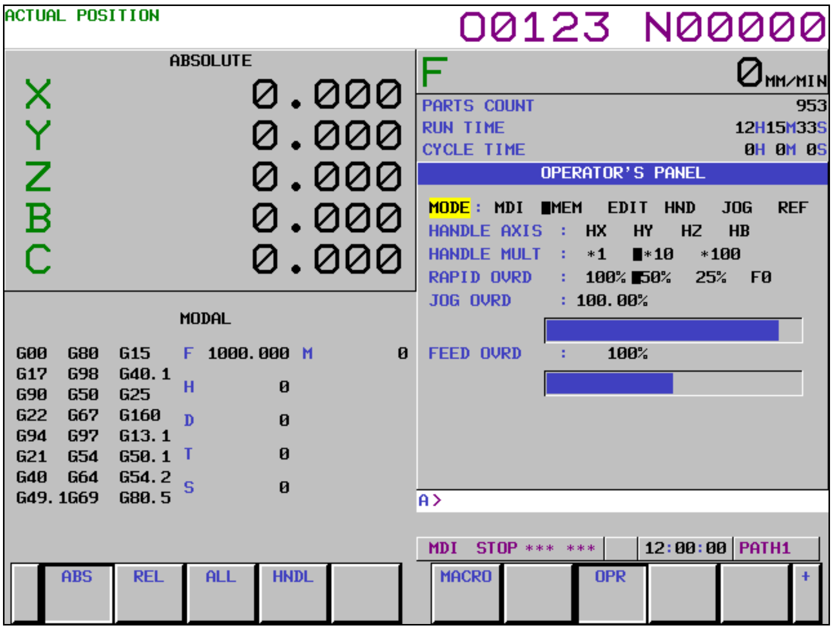


Fig. 12.3.8 (b) With the manual handle feed function

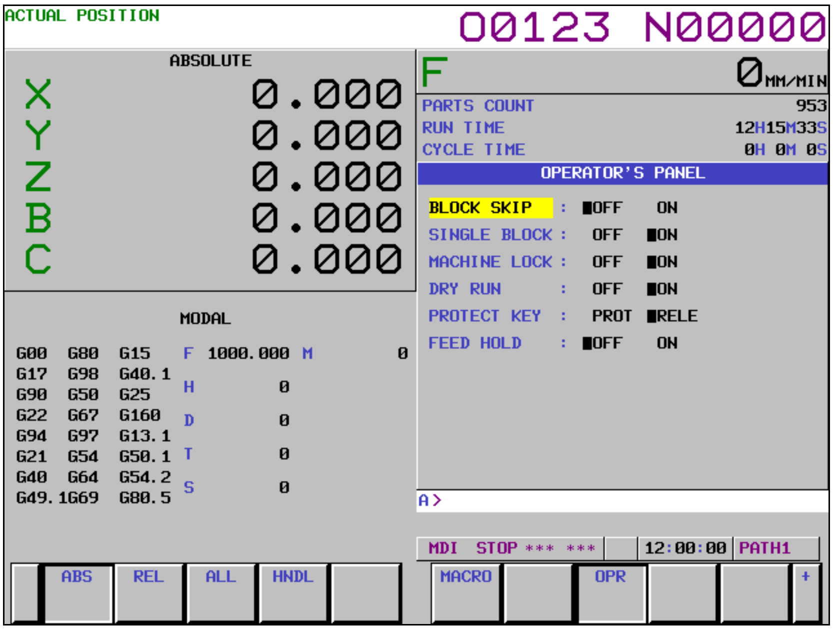


Fig. 12.3.8 (c)

4 Move the cursor to the desired switch by pressing cursor key [↑] or [↓].

5 Push the cursor key [←] or [→] to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform jog feed. Press the 5 key together with an arrow key to perform jog rapid traverse.

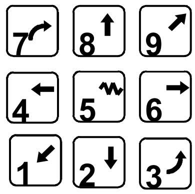


Fig. 12.3.8 (d) MDI arrow keys

Explanation

- Valid operations

The valid operations on the software operator's panel are shown below. Whether to use each group can be chosen using parameter No. 7200. Those groups that are not used are not displayed on the software operator's panel.

Group1 : Mode selection
Group2 : Selection of jog feed axis, Manual rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification
Group4 : Jog federate, federate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

- Screens on which jog feed is valid

When the LCD indicates other than the software operator's panel screen and self-diagnosis screen, jog feed is not conducted even if the arrow key is pushed.

- Jog feed and arrow keys

The feed axis and direction corresponding to the arrow keys can be set with parameters (Nos. 7210 to 7217).

- General purpose switches

For the meanings of these switches, refer to the manual issued by machine tool builder.

12.3.9 Setting and Displaying Tool Management Data

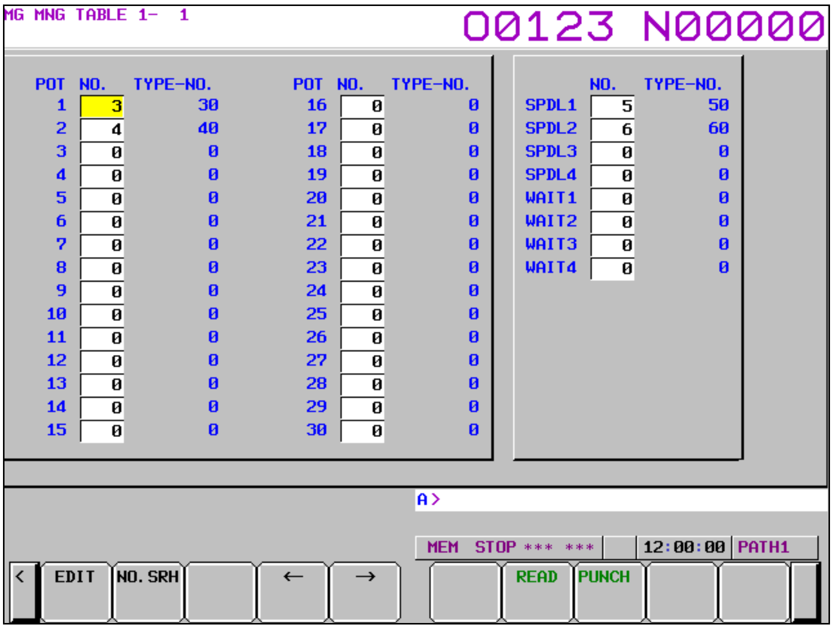
The tool management function totally manages tool information including tool offsets and tool life information.
This function provides a magazine screen and tool management screen.
This subsection explains how to set these screens.

12.3.9.1 Displaying and setting magazine screen

Procedure

1 Press the OFFSET SETTING function key.

2 Press the [TOOL MANAGER] chapter selection soft key.
Alternatively, press OFFSET SETTING several times until the tool management screen appears.
Press the [MAGAZINE] soft key.



Tool management magazine screen

3 By using the page keys, cursor keys, and [←] and [→] soft keys, move the cursor to the pot No. position of the magazine for which you want to set or modify data.
Alternatively, type the number of the tool management data you wan to set or modify, and press the [NO. SRH] soft key.

4 Press the [EDIT] soft key.

5 To set the tool management data number of a pot, type the tool management data number, then press the [INPUT] soft key.
To delete the tool management data number set for a pot, follow the steps below.
<1> Press the [ERASE] soft key.
<2> Press the [CUR DAT] soft key. To delete the tool management data numbers registered for all pots, press the [ALL] soft key.
<3> Press the [EXEC] soft key.
Alternatively, type 0.
6 To end the edit operation, press the [EXIT] soft key.
This returns the screen display to the conventional tool management screen.

Explanation

- Another method

Magazine data can be input/output also by using external I/O devices. See III-8, "DATA I/O".

- Displayed data

Pot : As many pots as specified in parameter No. 13222, 13227, 13232, or 13237 (settable for each magazine) are displayed. The starting pot number can be set for each magazine in parameter No. 13223, 13228, 13233, or 13238.

NO. : Tool management data numbers are displayed.
In the following cases, tools cannot be set in magazines:
1. A tool management data number beyond the range from 0 to (the number of valid pairs in parameter No. 13220) is set.
2. Tool management data is invalid (bit 1 of tool information is 0).
3. The tool management data number to be set is already set for another pot.

Type No.: Tool type numbers corresponding to tool management data numbers are displayed.
Values cannot be set on this screen.

Spindle : The tool management data numbers and tool type numbers of spindle positions are displayed.

Wait : The tool management data numbers and tool type numbers of wait positions are displayed.

- Edit mode

To edit data, press the [EDIT] soft key to enter tool management data edit mode.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. In addition, the number of tools used can be set in parameter No. 13220.

- Display of the spindle positions and wait positions

The spindle position and wait position can be set for each path.

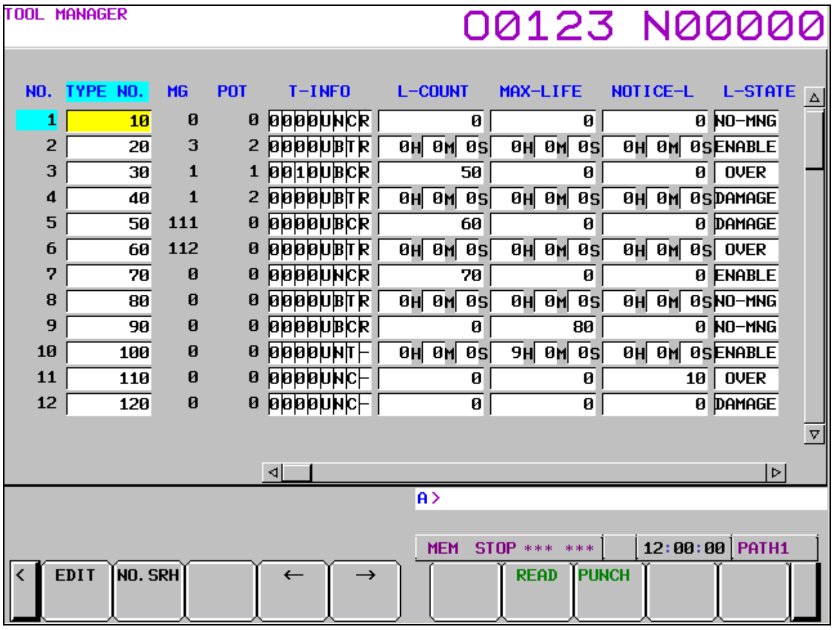
12.3.9.2 Displaying and setting tool management screen

Procedure

1 Press the OFFSET SETTING function key.

2 Press the [TOOL MANAGER] chapter selection soft key.
Alternatively, press OFFSET SETTING several times until the tool management screen appears.

3 Press the [TOOL MANAGER] soft key.



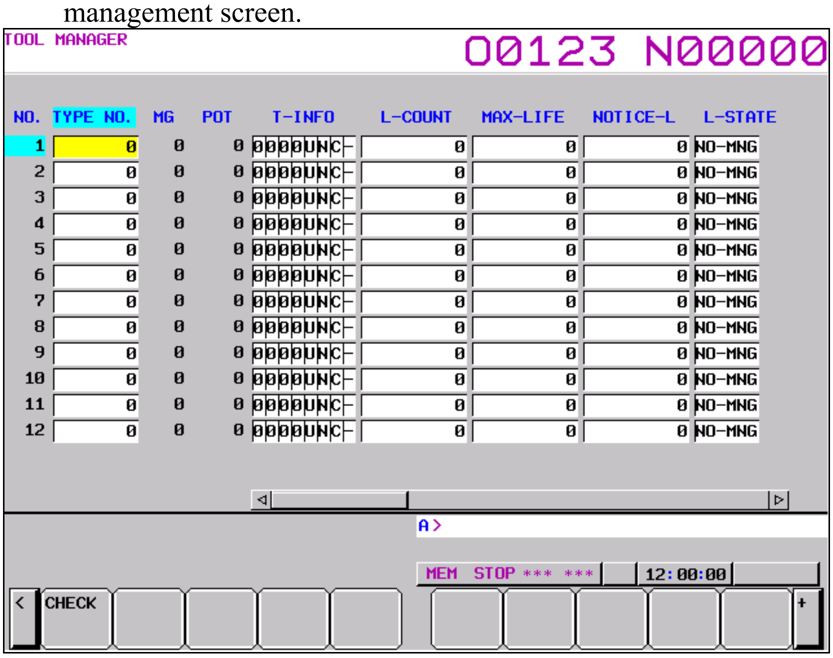
Tool management data screen

4 By using the page keys, cursor keys, and [←] and [→] soft keys, move the cursor to the position of the tool information of the tool number for which you want to set or modify data.
Alternatively, type the tool number of the data you want to set or modify, and press the [NO. SRH] soft key.

5 Press the [EDIT] soft key.

6 To set tool data, type a desired value, then press the [INPUT] soft key.
To delete tool information, follow the steps below.
<1> In step 4, move the cursor to the tool information you want to erase.
<2> Press the [ERASE] soft key.
<3> Press the [CUR DAT] soft key. To delete entire tool information, press the [ALL] soft key.
<4> Press the [EXEC] soft key.

7 To end the edit operation, press the [EXIT] soft key.
This returns the screen display to the conventional tool management screen.



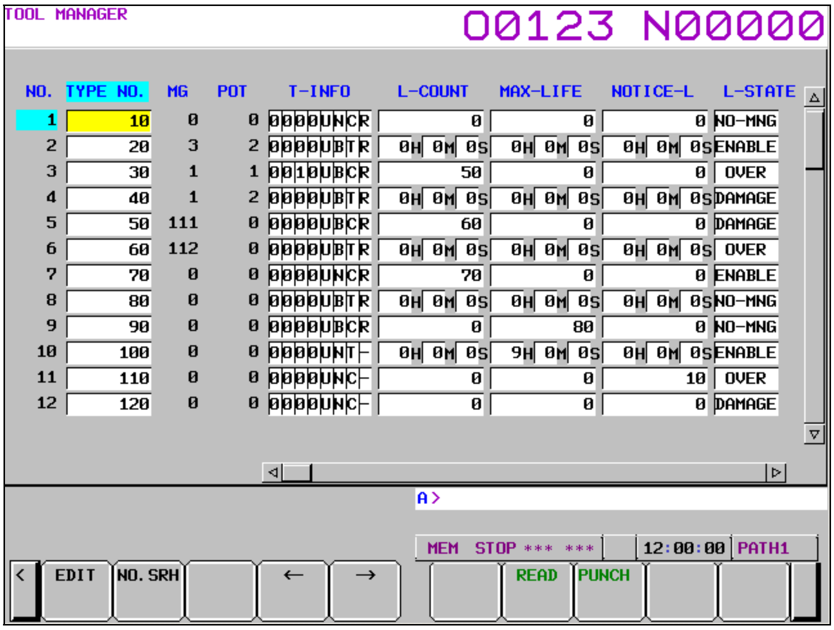
8 When soft key [CHECK] is pressed, if there are tools with the same number but with different count types (count and time), the cursor moves to the tool type number of the smallest tool management number in the tool type numbers and a warning message appears.
The warning messages issued by the check function are shown below.
<1> For the same type of tools with different count types
L-COUNT TYPE MISMATCH:XXXXXXXX
<2> For the same type of tools with the same count type
L-COUNT TYPE MATCH

Explanation
- Another method

Tool management data, customize data, and names set for tool states can be input/output also by using external I/O devices.
See III-8, "DATA I/O".

- Displayed information

- Life information



Tool management data life status screen

NO. : Tool management data numbers are displayed. These numbers can be displayed but cannot be set. The tool management data number of edited data is kept blinking until tool management data edit mode ends. Tool management data numbers set or deleted on the magazine screen also blink on the tool management screen.

Type No.: Tool type numbers are displayed.
A value from 0 to 99,999,999 can be set.

MG : The magazine number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

Pot : The pot number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

Tool information: The following five types of information are displayed sequentially from the right:

- Tool management data: Valid (R)/invalid (-))
- Life count type: Time (T)/count (C)
- Tool type: Large-diameter tool (B)/ normal tool (N)
- Data access: Locked (L)/unlocked (L)
- When a tool is not under life management: Included in the tools to be searched (1)/not included in the tools to be searched (0)

> **NOTE**
> 1 The tool types and data access information vary depending on the specifications defined by the machine tool builder.
> 2 The same type of tools must have the same life count type.

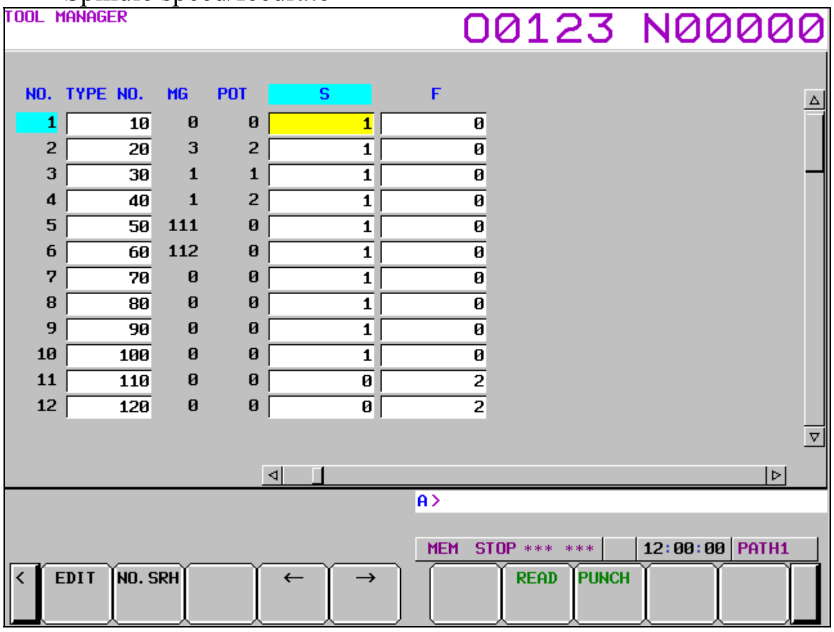
Life counter: The number of use times/use period of time of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

Maximum life: The maximum life value of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

Noticed life: Noticed life value of a tool
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

Life state: Current tool state
One of the four states, including invalid (0), present (1, 2), not present (3), and broken (4), is indicated. The numbers in parentheses are data values used when these states are input in MDI.

- Spindle speed/feedrate

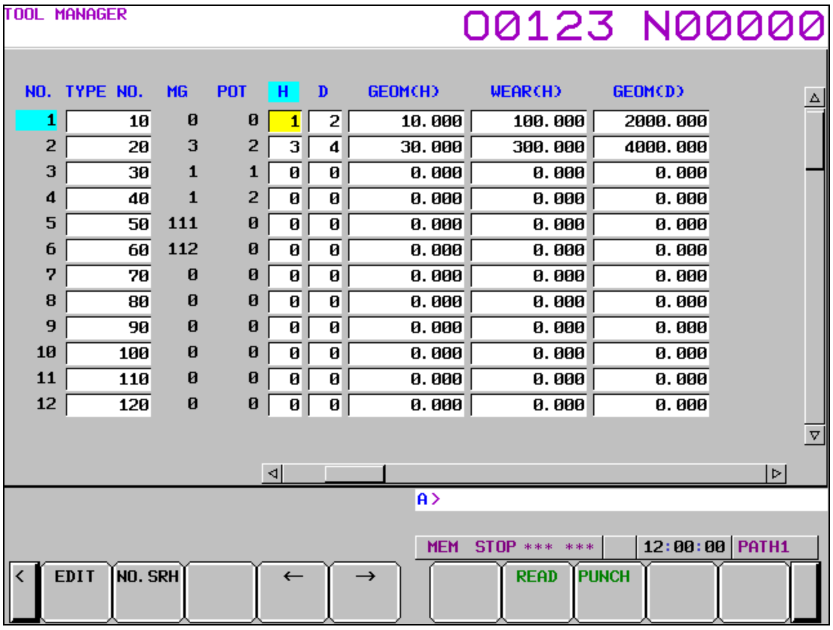


Tool management data spindle speed/feedrate screen

S : Spindle speed.
A value from 0 to 99,999 can be set.

F : Feedrate.
A value from 0 to 99,999,999 can be set.

- Tool offset information



Tool management data tool offset screen

H : Tool length compensation number (for machining center systems only).
A value from 0 to 999 can be set.

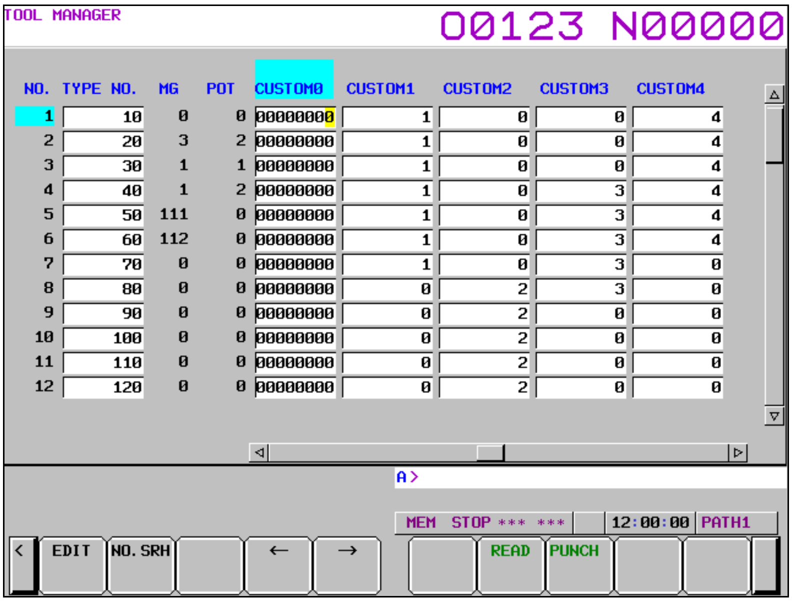
D : Cutter compensation number (for machining center systems only).
A value from 0 to 999 can be set.

TG : Tool geometry compensation number (for lathe systems only).
A value from 0 to 999 can be set.

TW : Tool wear compensation number (for lathe systems only).
A value from 0 to 999 can be set.

The displayed offset data is determined depending on the option configuration and parameter settings (bits 1(DCR), 2(DOY), 3(DOB), 4(DO2), 6(DOT), and 7(DOM) of parameter No. 13202).

- Customize information



NO.	TYPE NO.	MG	POT	CUSTOM0	CUSTOM1	CUSTOM2	CUSTOM3	CUSTOM4
1	10	0	0	00000000	1	0	0	4
2	20	3	2	00000000	1	0	0	4
3	30	1	1	00000000	1	0	0	4
4	40	1	2	00000000	1	0	3	4
5	50	111	0	00000000	1	0	3	4
6	60	112	0	00000000	1	0	3	4
7	70	0	0	00000000	1	0	3	0
8	80	0	0	00000000	0	2	3	0
9	90	0	0	00000000	0	2	0	0
10	100	0	0	00000000	0	2	0	0
11	110	0	0	00000000	0	2	0	0
12	120	0	0	00000000	0	2	0	0

Tool management data customize data screen

Customize 0 : Bit-type customize information.
For each bit, 1 or 0 can be input.

Customize 1 to 4 : Customize information. Any value from -99,999,999 to 99,999,999 can be set.

Customize 5 to 20 : Customize information. These items are displayed only when customize data extension option (5 to 20) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.

Customize 21 to 40 : Customize information. These items are displayed only when customize data extension option (5 to 40) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. The number of tools used can be set in parameter No. 13220.

- Edit mode

To edit data, press the [EDIT] soft key to enter tool management data edit mode.

- Life count override

When the period of time is set as the tool life count type, the life count can be overridden if bit 2 (LFV) of parameter No. 6801 is set to 1. Specify an override value by using a switch on the machine operator's panel.

Example) When cutting is performed for 10 minutes with an override of 0.1, one minute is counted in the tool life counter.

- Tool management extension function

When tool management extension functions are enabled, you can use the following functions in addition to the tool management functions:

- A value with a decimal point can be set as customize data.
  The maximum number of decimal places can be set to 7 by G10 data input and file read.
- Various types of tool management data can be protected using KEY signals.
  An attempt to enter a value to a protected data causes a warning message “WRITE PROTECT.”
- Tool life count period selection
  Bit 5 of tool information is used to make a life count period selection.

Item	Description
Data length	1 byte (flag data)
#5 REV	0: A life count period of 1 sec is used. 1: A life count period of 8 sec is used.

Range of count is as follows.

1sec : 0 to 3,599,999 seconds (999 hours 59 minutes 59 seconds)
8msec : 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

12.3.9.3 Each tool data screen

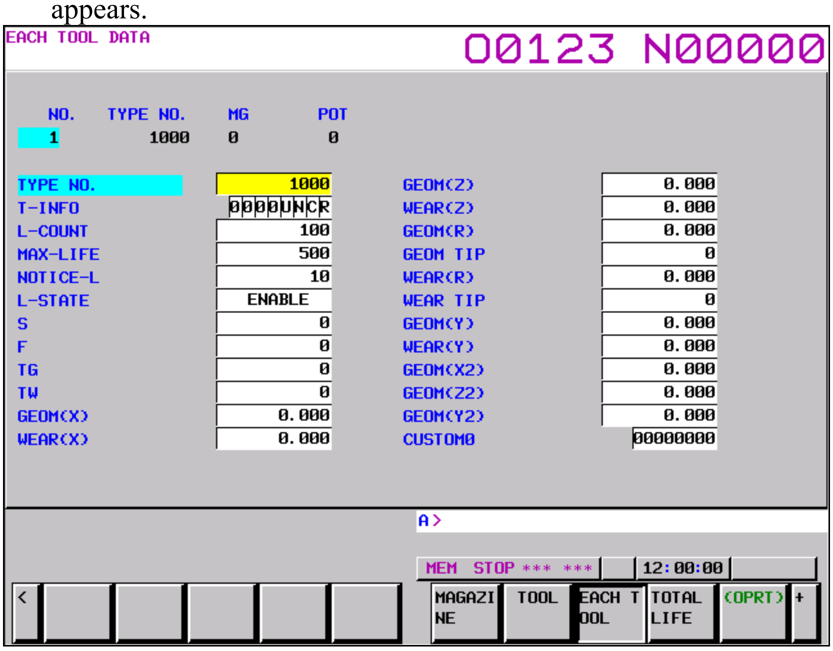
Each tool data screen

Procedure

1 Press the OFFSET SETTING function key.

2 Press the [TOOL MANAGER] chapter selection soft key.
Alternatively, press OFFSET SETTING several times until the tool management screen appears.

3 Press soft key [EACH TOOL]. The each tool data screen appears.



Each tool data screen

Explanation

- Header

The following four data items are displayed: NO., TYPE NO., MG, and POT.
When the data table of a tool extends over two or more pages, the same header is displayed on these pages.

- Data table

The data table shows data items related to a tool at a time.
The data items are displayed from the upper left to the lower left, the upper right, and the lower right in ascending order of screen display position numbers specified using the customize function. The number of digits displayed for one data item is fixed to 11.
Up to 24 data items are displayed on one page. (12 data items × 2 columns)
When more than 24 data items are set for a tool, the 25th and subsequent data items are displayed on the next page. (Up to three pages)

When a data item is set as a screen element of the tool management data screen twice or more using the tool management data display customize function (one of the tool management extension functions), only the data item with the smaller display position number is displayed. When no value is set for a data item, the field for the data item is not displayed and the next data item is displayed.

Key operations

- MDI key operations

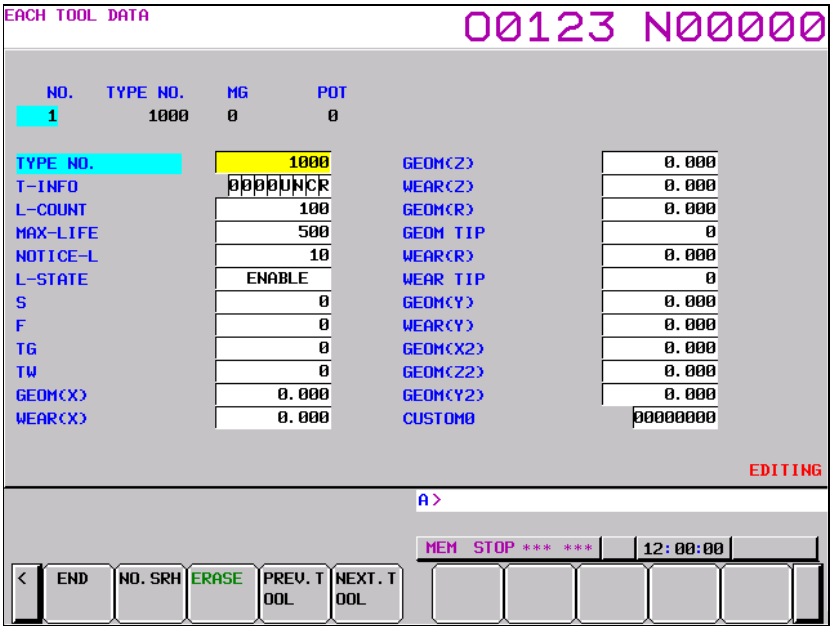
Numeral keys	Inputs a numeric value.
PAGE UP	Displays the previous page for the same tool.
PAGE DOWN	Displays the next page for the same tool.
<↑>	Moves the cursor to the previous data item. When the cursor is on the top row of the data table, it moves to the bottom row of the left column. When the cursor is on the first data item, it moves to the last data item.
<↓>	Moves the cursor to the next data item. When the cursor is on the bottom row of the data table, it moves to the top row of the right column. When the cursor is on the last data item, it moves to the first data item.
<←>	Moves the cursor left on the screen. When the cursor is on the left column of the data table, it moves to the right column on the row immediately above. When the cursor is on the first data item, it moves to the last data item.
<→>	Moves the cursor right on the screen. When the cursor is on the right column of the data table, it moves to the left column on the row immediately below. When the cursor is on the last data item, it moves to the first data item.

- Soft key operations

Soft key [EDIT]	Enters the management data edit mode.
Soft key [NO.SRH]	Displays data with a tool management data number when pressed after the tool management data number is entered.
Soft key [PREV.TOOL]	Returns to the previous tool management data number.
Soft key [NEXT.TOOL]	Proceeds to the next tool management number.

Operation in the management data edit mode

To edit data, press soft key [EDIT] to enter the management data edit mode.



In the management data edit mode, “EDITING” is displayed at the lower right of the screen.
In addition to the above key operations, the following key operations are available in the management data edit mode:

<INPUT>	Actually writes a value entered using numeric keys.
Soft key [END]	Ends the management data edit mode.

Operations for editing tool information are the same as those on the tool management data screen.

12.3.9.4 Displaying the total life of tools of the same type

Total life data screen

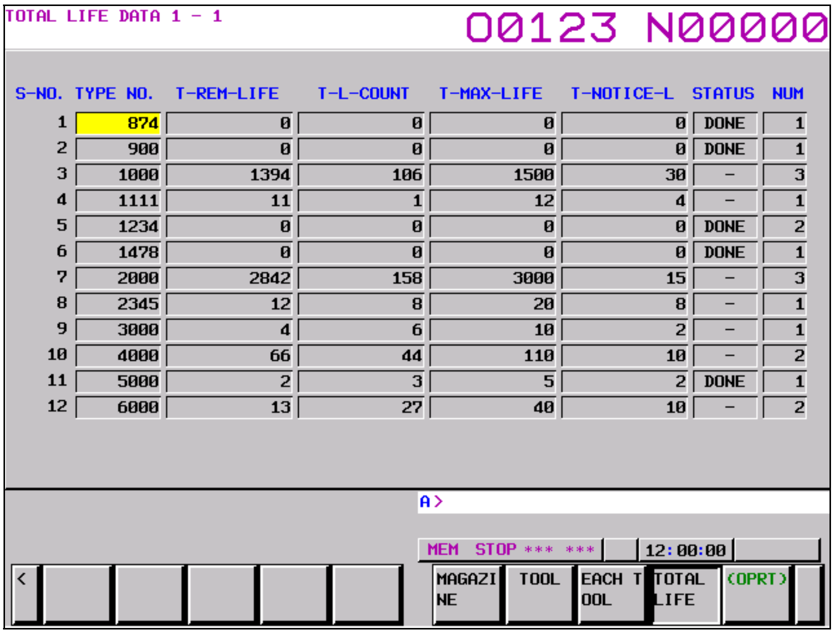
Procedure

1 Press the OFFSET SETTING function key.

2 Press the [TOOL MANAGER] chapter selection soft key.
Alternatively, press OFFSET SETTING several times until the tool management screen appears.

3 Press soft key [TOTAL LIFE]. The total life data screen appears.

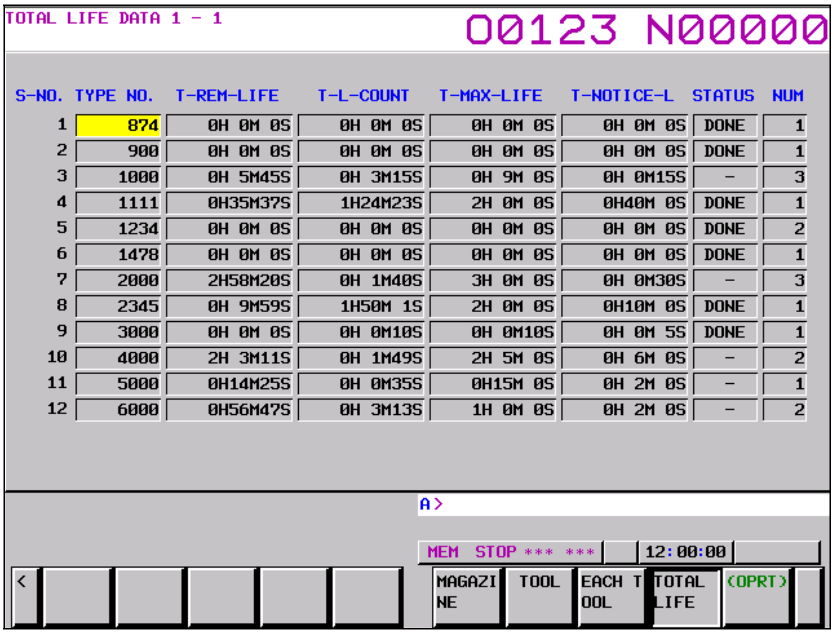
4 Press soft key [OPR], then soft key [CHANGE] to switch the display between total life data of the count counting type tools and that of the time counting type tools.


TOTAL LIFE DATA 1 - 1 O0123 N00000

S-NO.	TYPE NO.	T-REM-LIFE	T-L-COUNT	T-MAX-LIFE	T-NOTICE-L	STATUS	NUM
1	874	0	0	0	0	DONE	1
2	900	0	0	0	0	DONE	1
3	1000	1394	106	1500	30	-	3
4	1111	11	1	12	4	-	1
5	1234	0	0	0	0	DONE	2
6	1478	0	0	0	0	DONE	1
7	2000	2842	158	3000	15	-	3
8	2345	12	8	20	8	-	1
9	3000	4	6	10	2	-	1
10	4000	66	44	110	10	-	2
11	5000	2	3	5	2	DONE	1
12	6000	13	27	40	10	-	2

A>
MEM STOP *** *** 12:00:00
MAGAZINE TOOL EACH TOOL TOTAL LIFE (OPRT)

Count display



Time display

- Displayed information

S-NO.: Sequential number of each tool type

TYPE NO.: Tool type number

T-REM-LIFE: Total of remaining life values of tools with the same tool type number

T-L-COUNT: Total of used counts/times of tools with the same tool type number

T-MAX-LIFE: Total of maximum life values of tools with the same tool type number

NUM: Number of tools with the same tool type number

T-NOTICE-L: Total of notice life values of tools with the same tool type number when the tool life arrival signal is to be output for each tool type

STATUS: Whether the tool life arrival signal has been output when the signal is to be output for each tool type
Either of the two states (UNDONE and DONE) is displayed.

When bit 3 (ETE) of parameter No. 13200 is set to 0 and bit 2 (TRT) of parameter No. 13200 is set to 1, the tool life arrival signal is to be output for each tool type number and T-NOTICE-L and STATUS are displayed.

Key operations

- MDI key operations

PAGE UP	Displays the previous page. The cursor moves to the last data item on that page.
PAGE DOWN	Displays the next page. The cursor moves to the first data item on that page.
<↑>	Moves the cursor up on the screen.
<↓>	Moves the cursor down on the screen.
<←>	Disabled.
<→>	Disabled.

- Soft key operations

Soft key [DETAILS]	Displays the detailed life data screen.
Soft key [UPDATE]	Updates data displayed on the total life data screen.
Soft key [CHANGE]	Switch the display between the count and time types.
Soft key [TOOL TYPE SEARCH]	Moves the cursor to a tool type number when pressed after the number is entered.
Soft key [TOOL TYPE UP]	Sorts data in ascending order of tool type numbers.
Soft key [TOOL TYPE DOWN]	Sorts data in descending order of tool type numbers.
Soft key [REMAINING LIFE UP]	Sorts data in ascending order of remaining life values.
Soft key [REMAINING LIFE DOWN]	Sorts data in descending order of remaining life values.

NOTE

1. After soft key [TOOL TYPE UP], [TOOL TYPE DOWN], [REMAINING LIFE UP], or [REMAINING LIFE DOWN] is pressed, the cursor is positioned at the top of page 1 of the total life data screen.
2. When the power is turned on, data of the count counting type is displayed in ascending order of tool type numbers. When the display type is changed or data is sorted in a different order, the status is kept.
3. If soft key [DETAILS] is pressed when no data is displayed on the total life data screen, the warning message “NO DETAILED LIFE DATA SCREEN” is output.
4. If an unregistered tool type is specified and soft key [TOOL TYPE SEARCH] is pressed, the warning message “UNREGISTERED NUMBER” is output.

Detailed life data screen

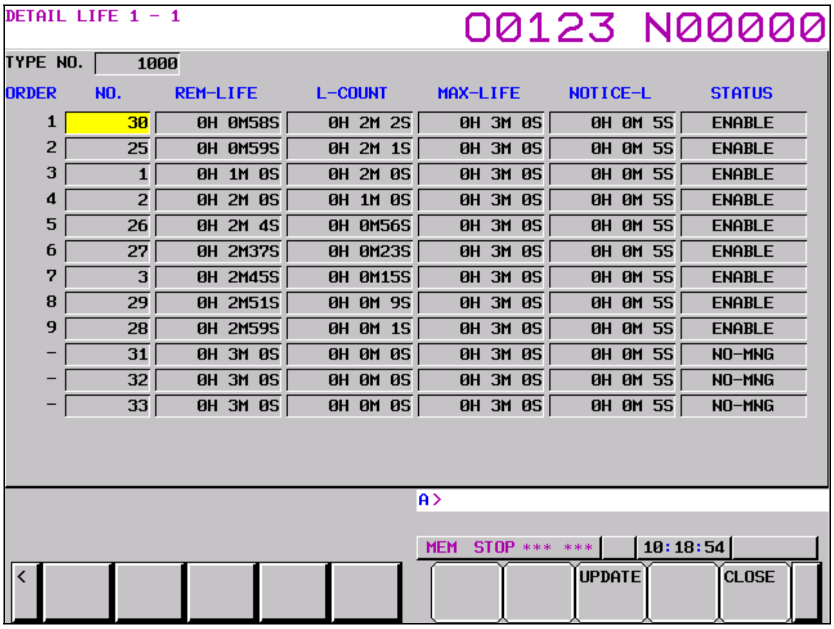
Procedure

1 Press the OFFSET SETTING function key.

2 Press the [TOOL MANAGER] chapter selection soft key. Alternatively, press OFFSET SETTING several times until the tool management screen appears.

3 Press soft key [TOTAL LIFE]. The total life data screen appears.

4 Press soft key [OPR], then soft key [DETAILS]. The detailed life data screen appears.



DETAIL LIFE 1 - 1 O0123 N00000

TYPE NO. 1000

ORDER	NO.	REM-LIFE	L-COUNT	MAX-LIFE	NOTICE-L	STATUS
1	30	0H 0M58S	0H 2M 2S	0H 3M 0S	0H 0M 5S	ENABLE
2	25	0H 0M59S	0H 2M 1S	0H 3M 0S	0H 0M 5S	ENABLE
3	1	0H 1M 0S	0H 2M 0S	0H 3M 0S	0H 0M 5S	ENABLE
4	2	0H 2M 0S	0H 1M 0S	0H 3M 0S	0H 0M 5S	ENABLE
5	26	0H 2M 4S	0H 0M56S	0H 3M 0S	0H 0M 5S	ENABLE
6	27	0H 2M37S	0H 0M23S	0H 3M 0S	0H 0M 5S	ENABLE
7	3	0H 2M45S	0H 0M15S	0H 3M 0S	0H 0M 5S	ENABLE
8	29	0H 2M51S	0H 0M 9S	0H 3M 0S	0H 0M 5S	ENABLE
9	28	0H 2M59S	0H 0M 1S	0H 3M 0S	0H 0M 5S	ENABLE
-	31	0H 3M 0S	0H 0M 0S	0H 3M 0S	0H 0M 5S	NO-MNG
-	32	0H 3M 0S	0H 0M 0S	0H 3M 0S	0H 0M 5S	NO-MNG
-	33	0H 3M 0S	0H 0M 0S	0H 3M 0S	0H 0M 5S	NO-MNG

A>

MEM STOP *** *** 10:18:54

UPDATE CLOSE

Detailed life data screen

- Displayed information

TYPE NO.:	Tool type number
ORDER:	Sequential number in ascending order of remaining life times or the order in which the customize data is set.
NO.:	Tool management data number
REM-LIFE:	Remaining life value obtained by subtracting the life count value from the maximum life value
L-COUNT:	Total used count or time of the tool
MAX-LIFE:	Maximum life value of the tool
NOTICE-L:	Notice life value of the tool
STATUS:	Life status of the tool One of the four states (NO-MNG (0), ENABLE (1, 2), NONE (3), and DAMAGED (4)) is displayed.

Key operations

- MDI key operations

PAGE UP	Displays the previous page.
PAGE DOWN	Displays the next page.
<↑>	Moves the cursor up on the screen. The cursor moves to the last data item on that page.
<↓>	Moves the cursor down on the screen. The cursor moves to the first data item on that page.
<←>	Disabled.
<→>	Disabled.

- Soft key operations

Soft key [UPDATA]	Updates data displayed on the detailed life data screen. The cursor is positioned at the beginning of page 1 after this soft key is pressed.
Soft key [CLOSE]	Closes the detailed life data screen and returns to the total life data screen.

NOTE

1. When soft key [CLOSE] is pressed and the total life data screen is displayed again, the cursor on the total life screen data is positioned as follows.
2. When the tools with the tool type number displayed when the detailed data screen is closed are registered as tool management data, the cursor on the total life screen is positioned at that tool type number.
3. When the tools with the tool type number displayed when the detailed data screen is closed are not registered as tool management data, the cursor on the total life data screen is positioned at the first total life data.

12.3.9.5 Tool geometry data screen

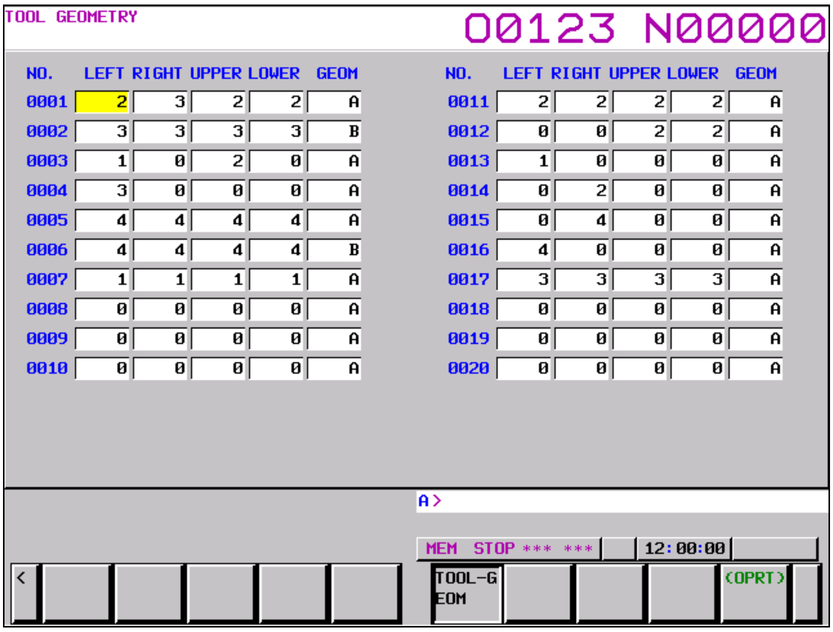
Tool geometry data screen

Procedure

1 Press the OFFSET SETTING function key.

2 Press the [TOOL MANAGER] chapter selection soft key.
Alternatively, press OFFSET SETTING several times until the tool management screen appears.

3 Press soft key [+], then soft key [TOOL-GEOM]. The tool geometry data screen appears.



Tool geometry data screen

- Displayed item

NO.: Tool geometry number
A value between 0 and 20 can be set.

LEFT: Sets the number of pots on the left of the reference pot that are to be occupied.
A value between 0 and 4 can be set.

RIGHT: Sets the number of pots on the right of the reference pot that are to be occupied.
A value between 0 and 4 can be set.

UPPER: Sets the number of pots above the reference pot that are to be occupied.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

LOWER: Sets the number of pots under the reference pot that are to be occupied.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

Key operations
- Operations in the standard mode
MDI key operations

Numeral keys	Inputs a numeric value.
<↑>	Moves the cursor up on the screen.
<↓>	Moves the cursor down on the screen.
<←>	Moves the cursor left on the screen.
<→>	Moves the cursor right on the screen.

Soft key operations

Soft key [NO.SRH]	Moves the cursor on a tool geometry number when pressed after the number is entered.
Soft key [EDIT]	Enters the data edit mode.
Soft key [REAL]	Reads data related to the tool management functions. This key is available only in the standard mode. Put the NC in the EDIT mode.
Soft key [PUNCH]	Punches data related to the tool management functions. This key is available only in the standard mode. Put the NC in the EDIT mode. In the management data edit mode, the following key operations are available in addition to the above key operations.

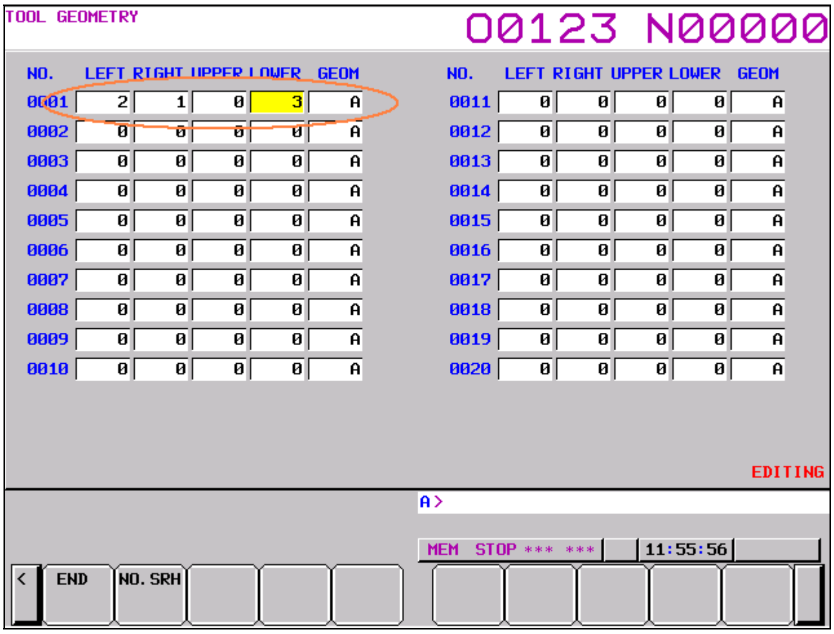
- Operations in the edit mode
To edit data, press soft key [EDIT] to enter the edit mode. In the edit mode, "EDITING" is displayed at the lower right of the screen.

MDI key operation
<INPUT> Actually writes a value entered using numeric keys.

Soft key operation
Soft key [END] Ends the data edit mode.

Example

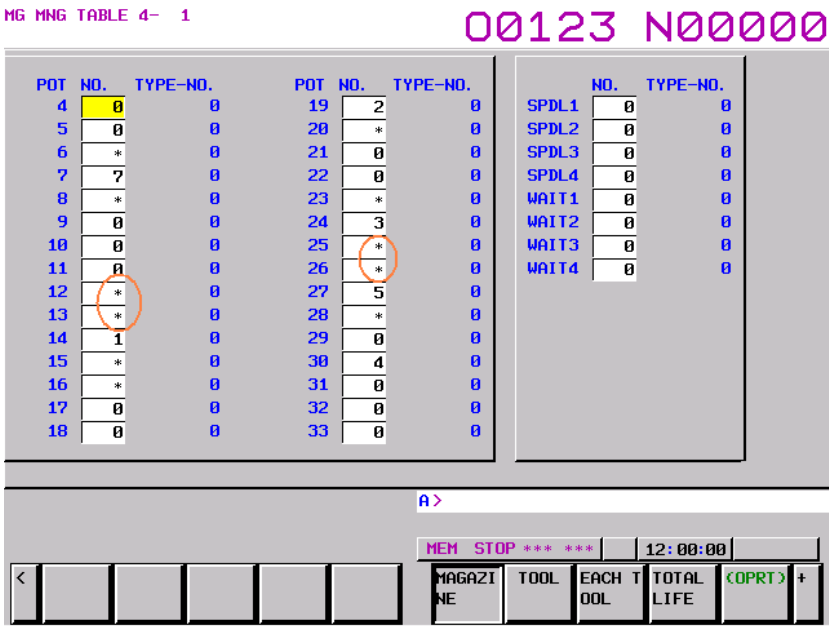
Set the edit mode. When the tool geometry with tool geometry number 1 occupies 1 pot in the left direction, 0.5 pots in the right direction, and 1.5 pots in the down direction, set data as shown in the figure below:



Example of setting data on the tool geometry data screen

- Display of occupied pots in the magazine management table

Each pot occupied by a tool stored in another pot is indicated with an asterisk (*).

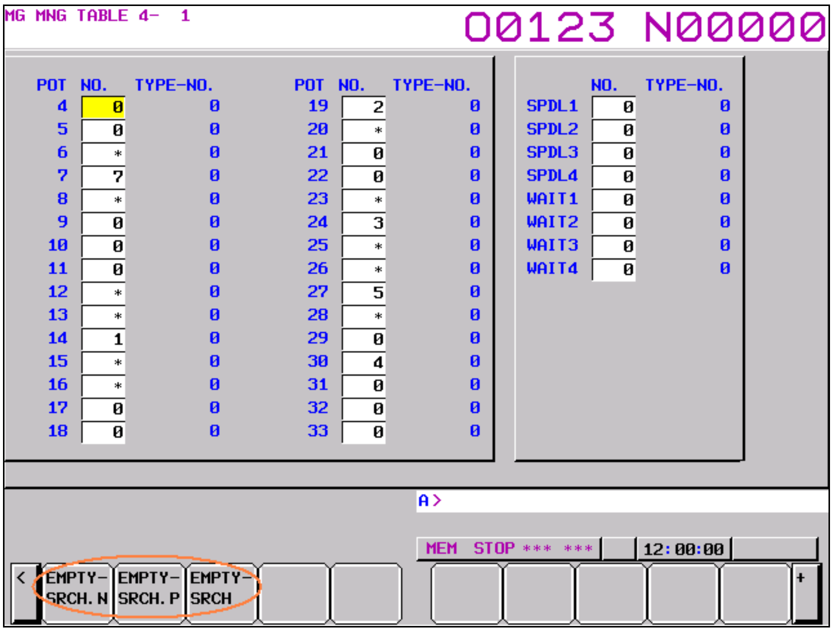


Magazine management table

If a tool to be registered for a magazine is determined to interfere with another tool, the warning message “TOOL INTERFERENCE CHECK ERROR:xxxx,xxxx” is displayed. xxxx indicates the tool number of each of the two tools. If a tool is determined to interfere with multiple tools, only the first tool detected by the NC interference check is displayed.
If a tool is determined to interfere with the frame of a magazine, the tool number of the tool to be stored and “FRAME” are displayed.

- Search for an empty pot for a large-diameter tool

In the management data edit mode, pressing soft key [OPR], then soft key [+] displays the soft keys for searching for an empty pot for a large-diameter tool.



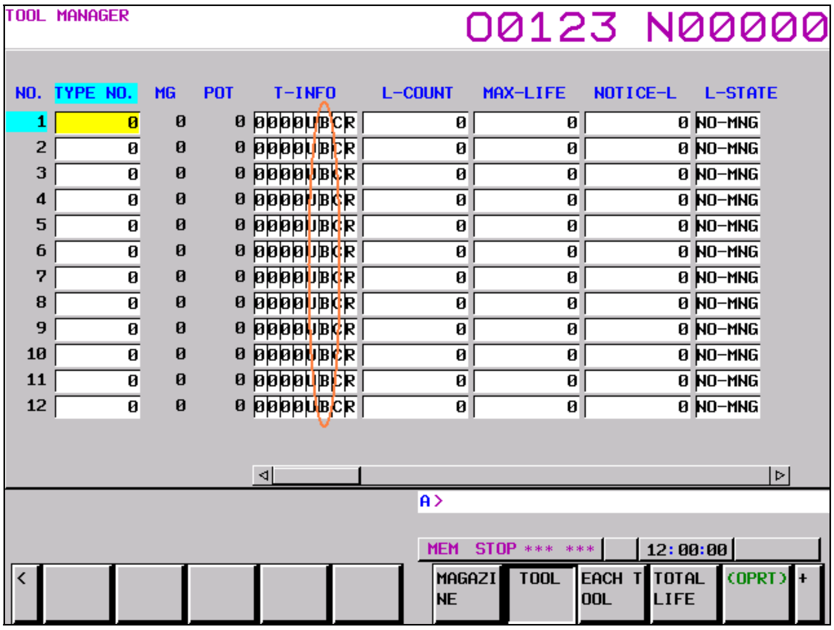
Searching for an empty pot for a large-diameter tool

Enter the tool geometry number in the key-in buffer and press a search soft key. The cursor moves to an empty pot fit for the geometry.

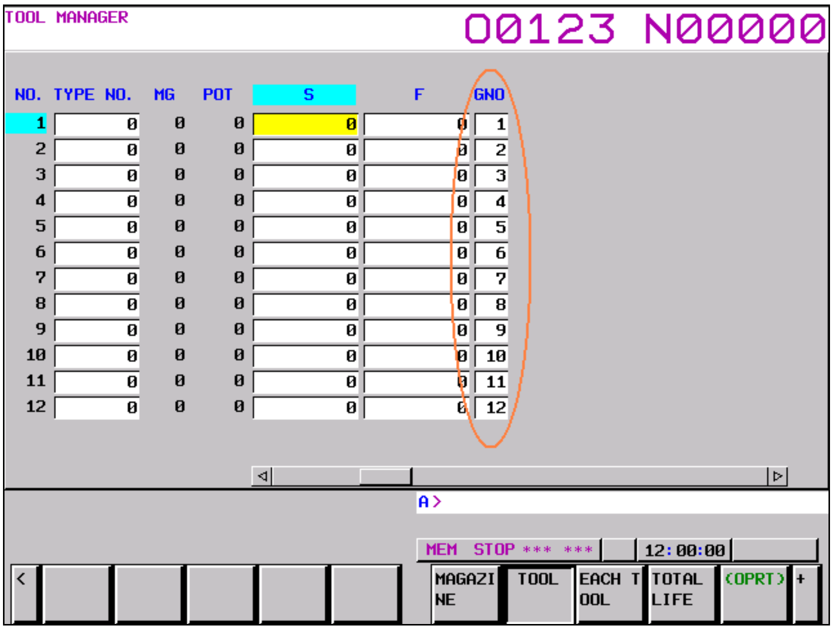
EMPTY-SRCH.N: Searches for an empty pot forward.
EMPTY-SRCH.P: Searches for an empty pot backward.
EMPTY-SRCH: Searches for the pot nearest to the current position.

- Tool management screen

You can use bit 2 of tool information to switch between a large-diameter tool and normal tool. For a large-diameter tool, set a tool geometry number fit for the tool.



Bit for switching between a normal tool and large-diameter tool



Tool geometry number

12.3.10 Displaying and Switching the Display Language

The language used for display can be switched to another language.
A display language can be set using a parameter. However, by modifying the setting of the display language on this screen, the display language can be switched without turning off then on the power.

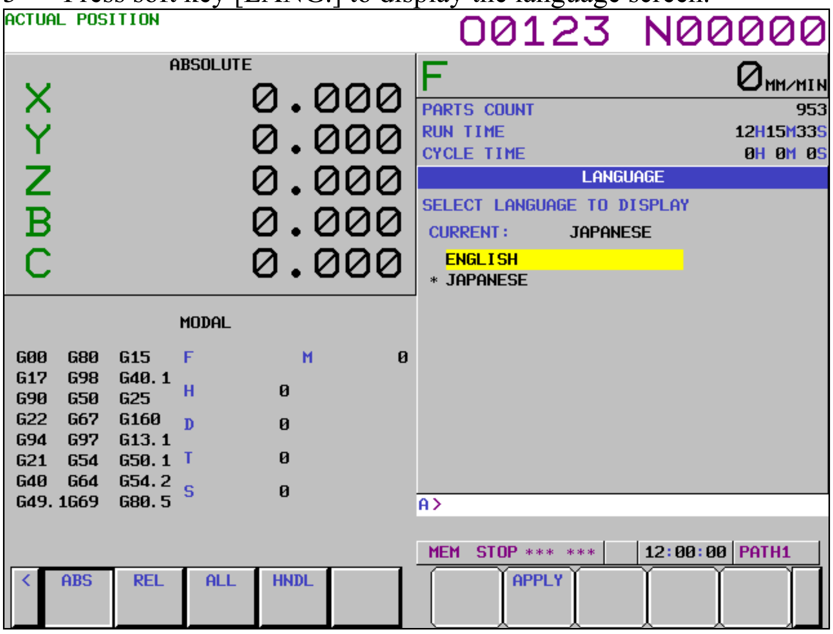
Displaying and setting the display language

Procedure

1 Press function key OFFSET SETTING .

2 Press the rightmost soft key (continuous menu key) several times.

3 Press soft key [LANG.] to display the language screen.



4 Press page key PAGE or PAGE, then press cursor keys , , , and/or to move the cursor to a desired display language.

5 Press soft key [APPLY]. The display language is switched to the selected language. The language specified on this screen continues to be used if the power is turned off then back on.

Explanation

- Language switching

The language screen can be displayed if bit 0 (NLC) of parameter No. 3280 is set to 0.

- Selectable languages

The display languages selectable on this screen are as follows:

1. English
2. Japanese
3. German
4. French
5. Chinese (Simplified)
6. Italian
7. Korean
8. Spanish
9. Dutch
10. Danish
11. Portuguese
12. Polish
13. Hungarian
14. Swedish
15. Czech

Among the languages listed above, English and other usable languages are displayed on the screen as a list of switchable languages.

Limitation

- Language parameter modification on the parameter screen

Which language to use for display is specified with parameter No. 3281. This parameter can be modified using the parameter screen as well. However, if a modification is made on the parameter screen, the new setting is not reflected until "APPLY" operation is performed on the language screen or the power is turned on again. If an invalid value is set in parameter No. 3281 on the parameter screen, the screen is displayed in English after the power is turned on again.

12.3.11 Protection of Data at Eight Levels

You can set eight CNC and PMC operation levels and one of eight protection levels for each type of CNC and PMC data.
When an attempt is made to change CNC and PMC data or output it to an external unit, the operation level is compared with the protection level to determine whether to allow the change or external output.

12.3.11.1 Operation level setting

You can set eight CNC and PMC operation levels.

Displaying and setting the operation level setting screen

Procedure

1 Press function key OFFSET SETTING .

2 Press the continuous menu key several times until [PROTECT] is displayed.

3 Press soft key [PROTECT].
The operation level setting screen shown below is displayed.

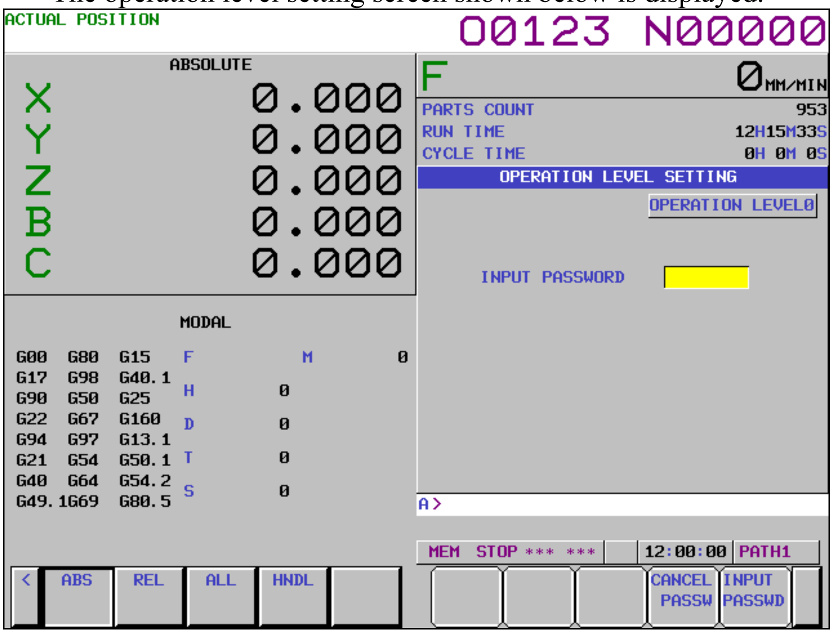


Fig. 12.3.11.1 (a) Operation level setting screen

4 Key in the password for an operation level to be set/modified, then press soft key [INPUT].

5 To return the operation level to 0, 1, 2, or 3, press soft key [CANCEL PASSWD].

Explanation
- Operation level setting

To select operation level 0 to 3, use the corresponding memory protection key signal.
To select operation level 4 to 7, use the corresponding password.

Table 12.3.11.1 (b) Operation level setting

Operation level	Setting	Sample grouping
7 (high)	Password	-
6	Password	MTB
5	Password	Dealer and integrator
4	Password	End user
3	Memory protection key signal	User level (level 1)
2	Memory protection key signal	User level (level 2)
1	Memory protection key signal	User level (level 3)
0 (low)	Memory protection key signal	User level (level 4)

When operation level 4 to 7 is set, the operation level remains unchanged until the password is cleared.
(The operation level also remains unchanged if the power is turned off.)
Operation level 7 is reserved for CNC and PMC maintenance.

> **NOTE**
> When a password is being entered, an asterisk (*) is displayed instead of each entered character.

12.3.11.2 Password modification

The current operation level is displayed.
The password for each of operation levels 4 to 7 can be modified.

Displaying and setting the password modification screen

Procedure

1 Press function key OFFSET SETTING .

2 Press the continuous menu key several times until [PROTECT] is displayed.

3 Press soft key [PROTECT].

4 Press soft key [PASSWORD].
The password change screen shown below is displayed.

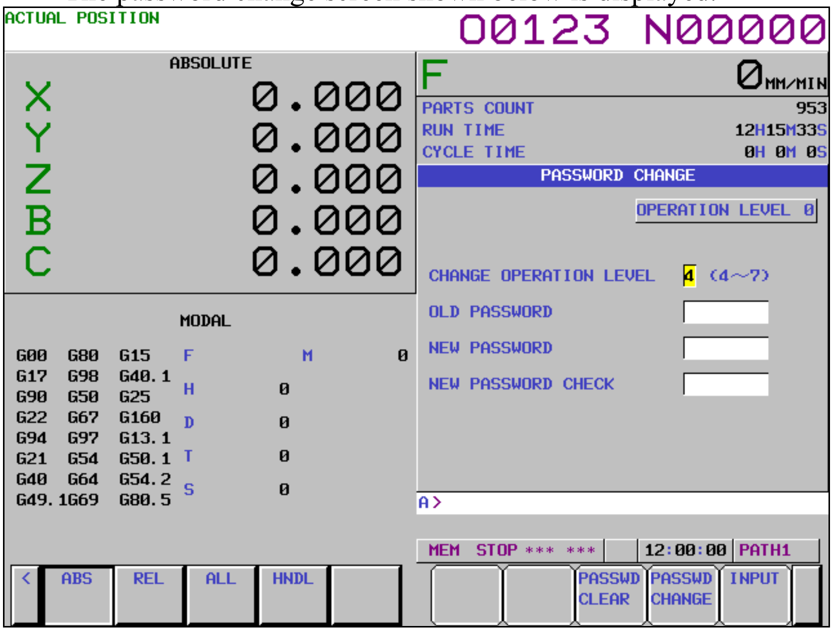


Fig. 12.3.11.2 (a) Password modification screen

5 Key in an operation level whose password is to be modified, then press soft key [INPUT].

6 Key in the current password for the operation level whose password is to be modified, then press soft key [INPUT].

7 Key in a new password, then press soft key [INPUT].

8 Key in the new password again for confirmation, then press soft key [INPUT].

9 Press soft key [PASSWDCHANGE].

10 To clear the password, press soft key [PASSWDCLEAR].

Explanation

Up to eight characters (only uppercase alphabetic characters and numeric characters) can be input.

> **NOTE**
> 1 For a password, consisting of three to eight characters, the following characters are available:
>   - Uppercase alphabetic characters
>   - Numeric characters
>
> 2 When a password is being entered, an asterisk (*) is displayed instead of each entered character.
>
> 3 Whether a password can be changed at the current operation level is determined as follows:
>   - Password of an operation level higher than the current operation level
>   Cannot be changed.
>   - Password of the current operation level
>   Can be changed.
>   - Password of an operation level lower than the current operation level
>   Can be changed (only to the initial password).
>
> 4 The set password is not displayed.
>   Be careful not to forget the password.

12.3.11.3 Protection level setting

The current operation level is displayed.
The change protection level and output protection level of each data item are displayed.
The change protection level and output protection level of each data item can be changed.

Confirmation based on protection level setting

Procedure

1 Press function key [OFFSET SETTING].

2 Press the continuous menu key [▷] several times until [PROTECT] is displayed.

3 Press soft key [PROTECT].

4 Press soft key [DATA LEVEL] to change the protection level of CNC data or press soft key [PMC LEVEL] to change the protection level of PMC data.
The following protection level change screen is displayed.

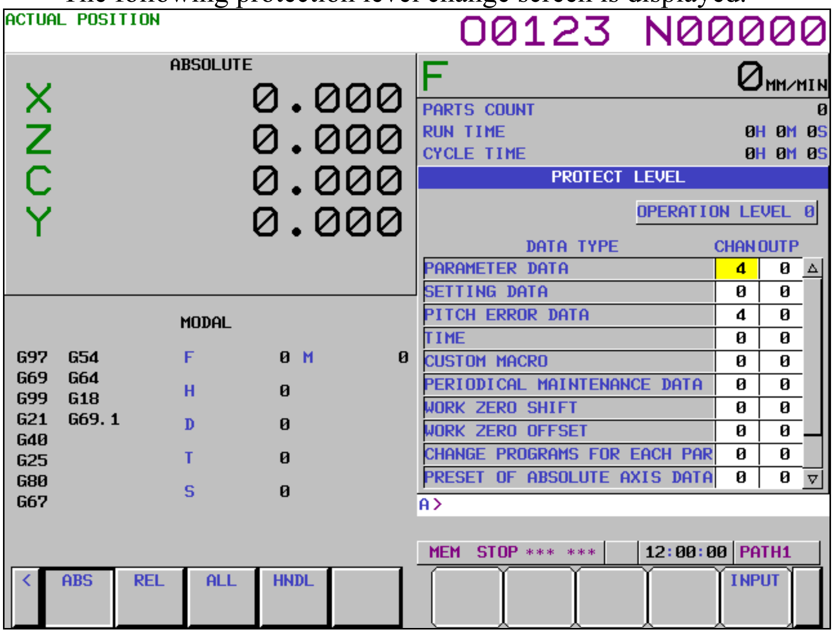


Fig. 12.3.11.3 (a) Protection level change screen

5 Move the cursor to the change level or output level of a desired data item.

6 Key in a new desired level, then press soft key [INPUT].

> **NOTE**
> When the protection level of PMC data is set, soft key [PMC SWITCH] is used to switch between PMC paths to be set, for multipath PMC.

Explanation

When the protection level of a data item is higher than the current operation level, the protection level of the data item cannot be changed.
The protection level of a data item cannot be changed to a protection level higher than the current operation level.
For each of the following types of data, you can set a data protection level.
There are the following two types of data protection levels:

- Change protection level
  Sets the protection level used when data is changed.
- Output protection level
  Sets the protection level used when data is output (punched out) to an external unit.

As a protection level, you can set a value of 0 (low) to 7 (high).

Table 12.3.11.3 (c) Protection level of each type of data

Type of data	Initial protection level	
	Change	Output
Custom macro variable data <CUSTOM MACRO> (including variable data dedicated to the macro executor)	0	0
Periodical maintenance data <PERIODICAL MAINTENANCE DATA>	0	0
Tool management data <"TOOL LIFE TIME DATA>	0	0
Tool offset data <TOOL OFFSET DATA> (For each type when tool geometry compensation and tool wear compensation are treated differently)	0	0
Clock data <TIME>	0	0
Workpiece origin shift amount data <WORK ZERO SHIFT>	0	0
Workpiece origin offset data <WORK ZERO OFFSET>	0	0
Parameter data <PARAMETER DATA>	4	0
Settings <SETTING DATA>	0	0
Pitch error compensation data <PITCH ERROR DATA> (including three-dimensional error compensation data)	4	0
Each part program <PROGRAMS FOR EACH PARTS>	0	0
Part program editing operation <CHANGE PROGRAMS FOR EACH PARTS>	0	0
Absolute coordinate preset operation <PRESET OF ABSOLUTE AXIS DATA>	0	0

Table 12.3.11.3 (d) Protection level of PMC data

Type of data	Initial protection level	
	Change	Output
Composition parameter	0	0
Setting (online)	0	0
Setting (each path)	0	0
Sequence program	0	0
PMC parameter	0	0
Timer	0	0
Counter	0	0
Keep relay	0	0
Keep relay (system)	0	0
Data table	0	0
Data table control	0	0
PMC momory	0	0

NOTE
1. For some types of data, the output function is not provided.
2. When the protection level of data is higher than the current operation level, the protection level cannot be changed.
3. The protection level of data cannot be changed to a level higher than the current operation level.
4. Settable types of data increase or decrease, depending on the option configuration.
5. For details on the protection level of PMC data, refer to "PMC Programming Guide(B-63983EN)".

12.3.11.4 Setting the change protection level and output protection level of a program

The display/operations indicated below can be performed from the directory screen.
The change protection level and output protection level of each part program are displayed.
The change protection level and output protection level of each part program can be changed.

Setting the change protection level and output protection level of a program

Procedure

1 Press function key PROG .

2 Press soft key [LIST].
The program directory screen shown below is displayed.

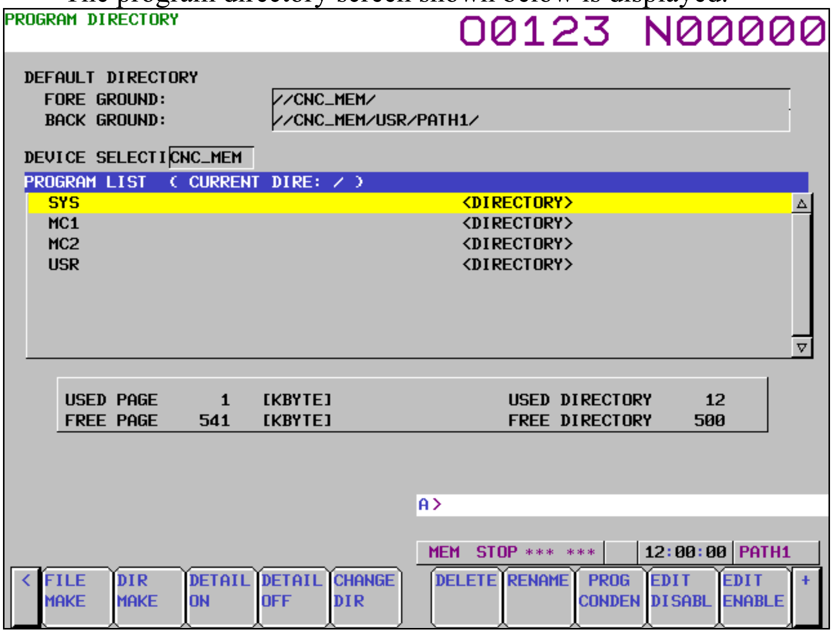


Fig. 12.3.11.4 (a) Program directory screen

3 Press the continuous menu key several times until [DETAILON] is displayed.

4 Press soft key [DETAILON]. The screen display switches to the detail display screen.

5 Move the cursor to a desired program.

6 Press the continuous menu key several times until [CHANGELEVEL] is displayed.

7 Key in a new desired level, then press soft key [CHANGELEVEL].

8 To change the output protection level, key in a new desired level, then press soft key [OUT LEVEL].

Explanation

The change protection level (0 to 7) and output protection level (0 to 7) are displayed as "CHANGE PROTECTION LEVEL VALUE/OUTPUT PROTECTION LEVEL".

NOTE

1. When the protection level of data is higher than the current operation level, the protection level cannot be changed.
2. The protection level of data cannot be changed to a level higher than the current operation level.
3. A protection level can be set only for those part programs that are held on the "CNC_MEM" device.

12.3.12 Precision Level Selection

An intermediate precision level between the parameters for emphasis on velocity (precision level 1) and the parameters for emphasis on precision (precision level 10) set on the machining parameter tuning screen can be selected. As shown in the figure below, the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

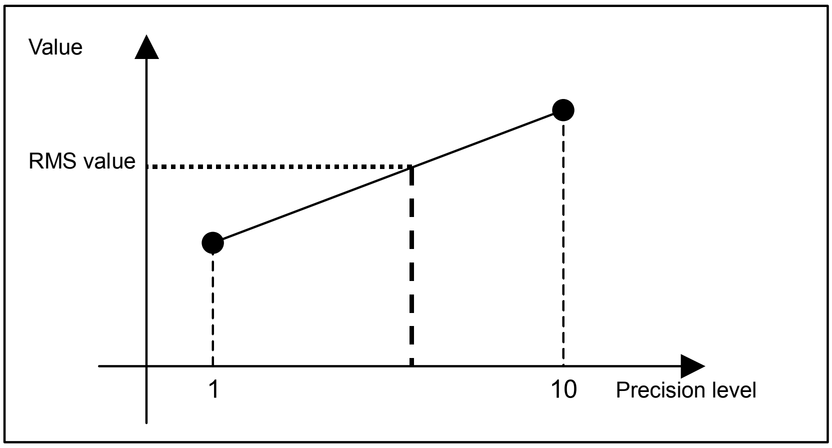


Fig. 12.3.12 (a) Image of "level"

Procedure for precision level selection

1 Set the MDI mode.

2 Press function key OFFSET SETTING .

3 Press soft key [PR-LEV].

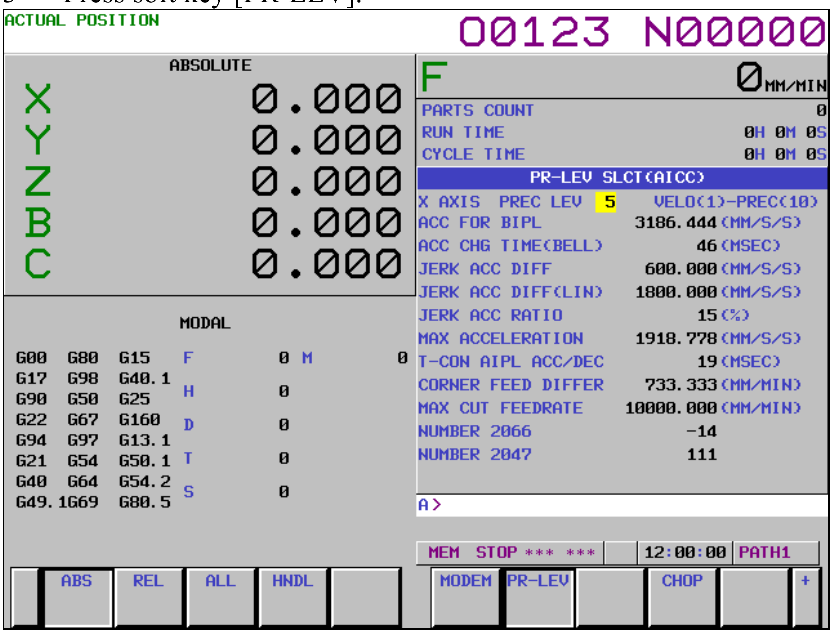


Fig. 12.3.12 (b) Precision level selection screen

4 To change the precision level, key in a desired precision level (1 to 10), then press the INPUT key on the MDI panel.

5 When the precision level is changed, a RMS value is obtained from the velocity-emphasized parameter set and precision-emphasized parameter set for parameter modification. For the modified parameters, see the description of the machining parameter tuning.

6 If there is an axis in addition to the currently displayed axes, press page key PAGE ↑ or PAGE ↓ several times to display the screen for the axis.

12.4 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM

When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.
This chapter describes how to set parameters on the MDI panel. Parameters can also be set with external input/output devices such as the memory card (see III-8).
In addition, pitch error compensation data used for improving the precision in positioning with the ball screw on the machine can be set or displayed by the operations under function key SYSTEM .
See III-7 for the diagnosis screens displayed by pressing function key SYSTEM .

12.4.1 Displaying and Setting Parameters

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.
Normally, the user need not change parameter setting.

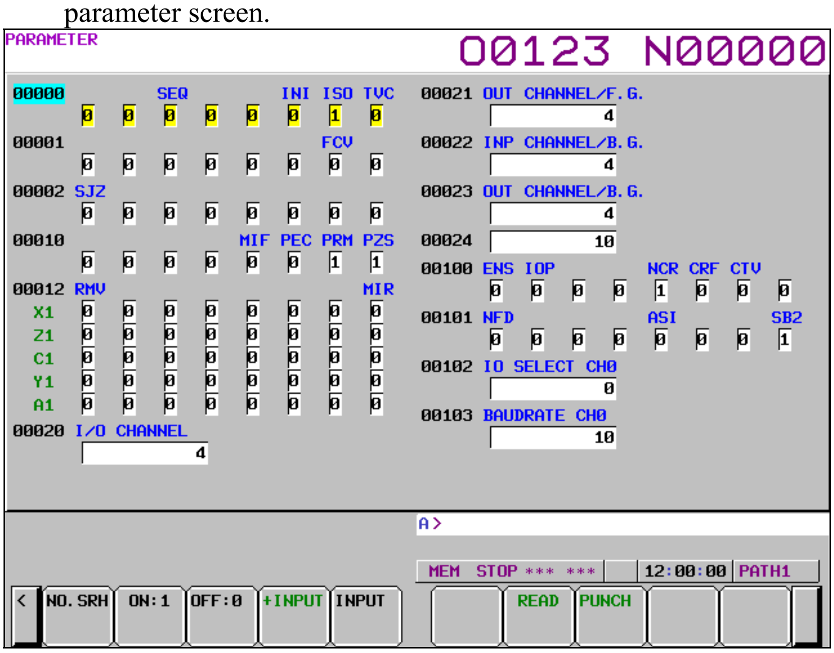
Procedure for displaying and setting parameters

Procedure

1 Set 1 for PARAMETER WRITE to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key SYSTEM .

3 Press chapter selection soft key [PARAM] to display the parameter screen.



4 Move the cursor to the parameter number to be set or displayed in either of the following ways:
- Enter the parameter number and press soft key [NO.SRH] .
- Move the cursor to the parameter number using the page keys, PAGE ↑ and PAGE ↓, and cursor keys, ↑, ↓, ←, and →.

5 To set the parameter, enter a new value with numeric keys and press soft key [INPUT]. The parameter is set to the entered value and the value is displayed.

6 Set 0 for PARAMETER WRITE to disable writing.

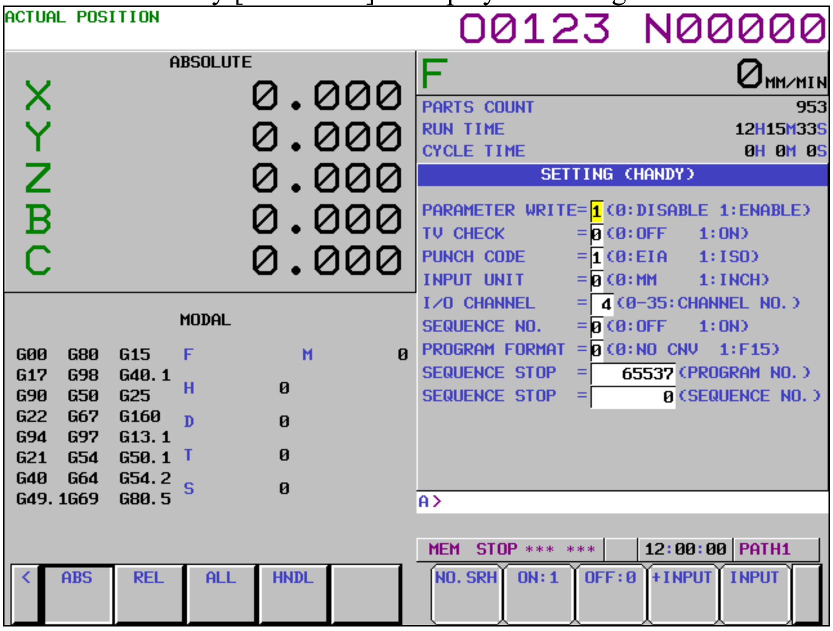
Procedure for enabling/displaying parameter writing

Procedure

1 Select the MDI mode or enter state emergency stop.

2 Press function key OFFSET SETTING .

3 Press soft key [SETTING] to display the setting screen.



4 Move the cursor to PARAMETER WRITE using cursor keys.

5 Press soft key [(OPRT)], then press [ON :1] to enable parameter writing.
At this time, the CNC enters the alarm state (SW0100).

6 After setting parameters, return to the setting screen. Move the cursor to PARAMETER WRITE and press soft key [(OPRT)] , then press [OFF :0].

7 Depress the RESET key to release the alarm condition.
When an alarm (PS0000) occurred, however, it is not released unless the power is turned off and back on.

Explanation

- Setting parameters with external input/output devices

See III-8 for setting parameters with external input/output devices such as the memory card.

- Parameters that require turning off the power

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm PW0000. In this case, turn off the power, then turn it on again.

- Parameter list

Refer to the Parameter Manual (B-63950EN) for the parameter list.

- Setting data

Some parameters can be set on the setting screen if the parameter list indicates "Setting entry is acceptable". Setting 1 for PARAMETER WRITE is not necessary when three parameters are set on the setting screen.

12.4.2 Displaying and Setting Pitch Error Compensation Data

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.

Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.

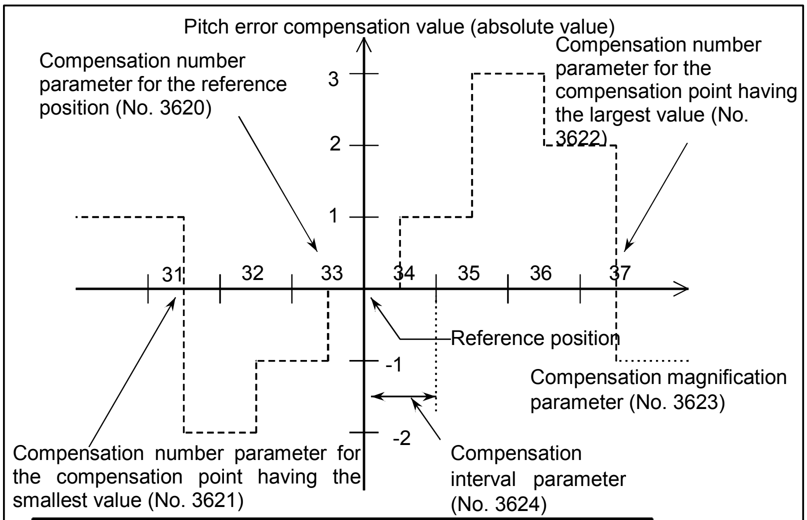
The pitch error compensation data is set according to the characteristics of the machine connected to the NC. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced.

In principle, the end user must not alter this data.

Pitch error compensation data can be set with external devices such as the memory card (see Chapter III-8). Compensation data can also be written directly with the MDI panel.

The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation point number set by these parameters.

In the following example, 33 is set for the pitch error compensation point at the reference position.



Compensation position number	31	3	3	3	3	3	3
Compensation value to be set	-3	+1	+1	+1	+2	-1	-3

- Number of the pitch error compensation point at the reference position (for each axis): Parameter 3620
- Number of the pitch error compensation point having the smallest value (for each axis): Parameter 3621
- Number of the pitch error compensation point having the largest value (for each axis): Parameter 3622
- Pitch error compensation magnification (for each axis): Parameter 3623
- Interval of the pitch error compensation points (for each axis): Parameter 3624

- Travel distance per revolution of pitch error compensation of the rotary axis type (for each axis): Parameter 3625

- Bi-directional pitch error compensation

The bi-directional pitch error compensation function allows independent pitch error compensation in different travel directions. (When the movement is reversed, compensation is automatically carried out as in a backlash.)
To use this function, specify pitch error compensation for each travel direction, that is, separately for the positive and negative directions of a movement.
When using bi-directional pitch error compensation (setting the parameter BDP (No. 3605#0) to 1), specify the following parameters in addition to the pitch error compensation parameter.

- Number of the pitch error compensation point at the negative end (for travel in the positive direction, for each axis): Parameter 3621
- Number of the pitch error compensation point at the positive end (for travel in the positive direction, for each axis): Parameter 3622
- Number of the pitch error compensation point at the negative end (for travel in the negative direction, for each axis): Parameter 3626
- Pitch error compensation in the reference position when moving to the reference position from opposite to the reference position return direction (for each axis) Parameter 3627

Procedure for displaying and setting the pitch error compensation data

Procedure

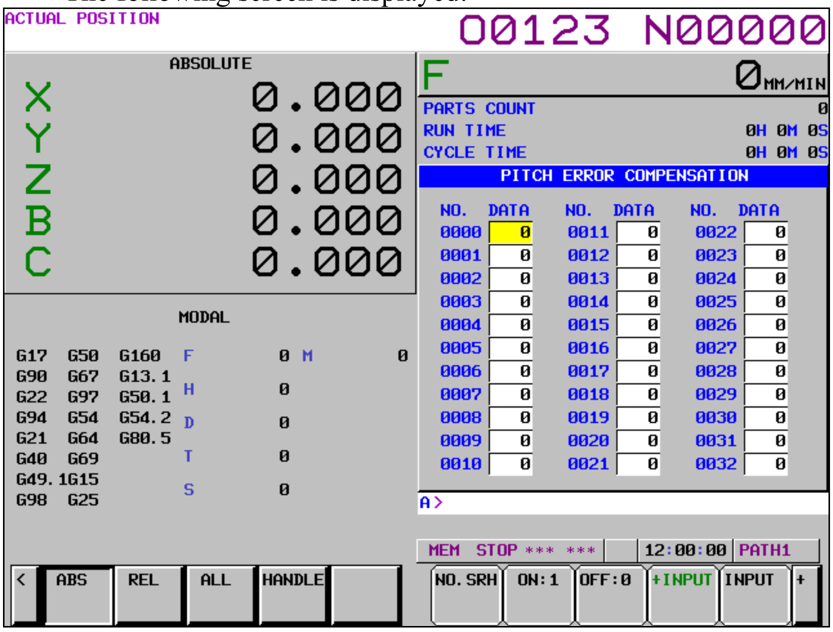
1 Set the following parameters:
   - Number of the pitch error compensation point at the reference position (for each axis): Parameter 3620
   - Number of the pitch error compensation point having the smallest value (for each axis): Parameter 3621
   - Number of the pitch error compensation point having the largest value (for each axis): Parameter 3622
   - Pitch error compensation magnification (for each axis): Parameter 3623
   - Interval of the pitch error compensation points (for each axis): Parameter 3624
   - Travel distance per revolution of pitch error compensation of the rotary axis type (for each axis): Parameter 3625

   When using bi-directional pitch error compensation (setting the parameter BDP (No. 3605#0) to 1), specify the following parameters in addition to the pitch error compensation parameter.
   - Number of the pitch error compensation point at the negative end (for travel in the positive direction, for each axis): Parameter 3621

- Number of the pitch error compensation point at the positive end (for travel in the positive direction, for each axis): Parameter 3622
- Number of the pitch error compensation point at the negative end (for travel in the negative direction, for each axis): Parameter 3626
- Pitch error compensation in the reference position when moving to the reference position from opposite to the reference position return direction (for each axis) Parameter 3627

2 Press function key SYSTEM .

3 Press the continuous menu key ▷ , then press chapter selection soft key [PITCH].
The following screen is displayed:



4 Move the cursor to the compensation point number to be set in either of the following ways:

- Enter the compensation point number and press the [NO.SRH] soft key.
- Move the cursor to the compensation point number using the page keys, PAGE ↑ and PAGE ↓, and cursor keys, ↑, ←, ↓, and →.

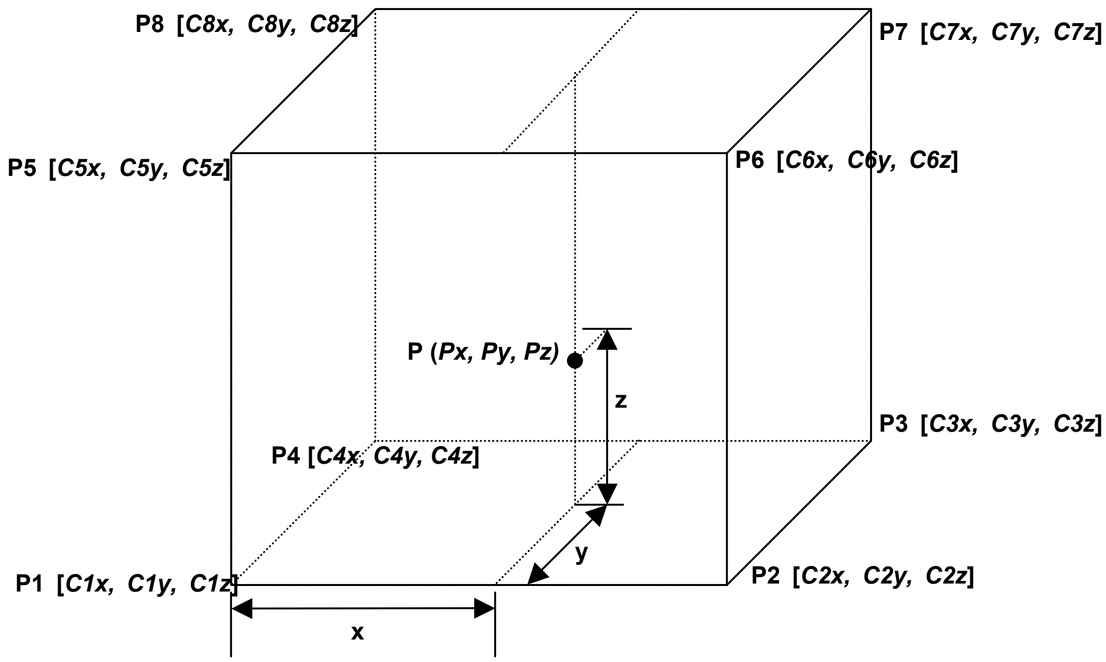
5 Enter a value with numeric keys and press the [INPUT] soft key.

12.4.3 Displaying and Setting Three-Dimensional Error Compensation Data

In ordinary pitch error compensation, compensation is applied to a specified compensation axis (single axis) by using its position information. For example, pitch error compensation is applied to X-axis by using the position information of X-axis.
Three-dimensional error compensation is a function that adjusts the current position by calculating compensation data (for three axes) from the compensation amounts at surrounding compensation points (eight points) on the basis of the interior division ratio in the compensation area (rectangular parallelepiped) containing the current position on up to three compensation axes.

- Calculation of compensation

Three-dimensional error compensation is calculated as follows.



Let three compensation axes be X, Y, and Z (three basic axes) and the coordinates of the current position be P (Px, Py, Pz). consider a compensation space (rectangular parallelepiped) containing P. Let its vertexes be P1, P2, …, and P8 and the compensation values for the individual axes at the individual vertexes be Cnx, Cny, and Cnz (where n is a number between 1 and 8).
Let the interior division ratio on X-axis at P be x. Here, x is standardized in the range of 0 to 1 as follows:

$$x = \frac{|Px - P1x|}{|P2x - P1x|}$$

P1x and P2x are the X coordinates of P1 and P2. The interior division ratios on Y and Z-axes are determined in the same way.

The compensation amount Cx for X-axis at P isdetermined as follows:

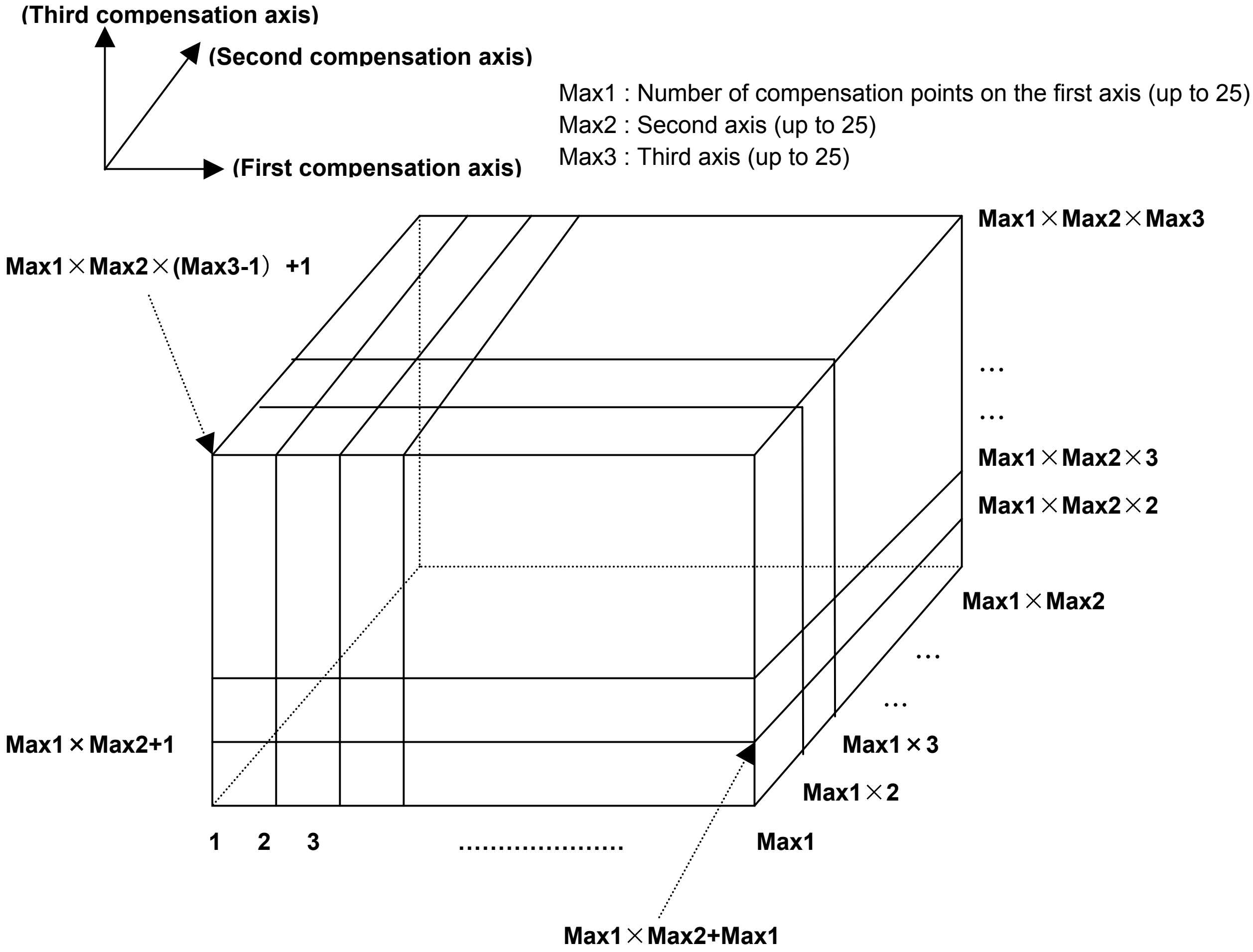
$$\begin{aligned}Cx = {} & C1x\times(1-x)\times(1-y)\times(1-z)+C2x\times x\times(1-y)\times(1-z)\\ & +C3x\times x\times y\times(1-z)+C4x\times(1-x)\times y\times(1-z)\\ & +C5x\times(1-x)\times(1-y)\times z+C6x\times x\times(1-y)\times z\\ & +C7x\times x\times y\times z+C8x\times(1-x)\times y\times z\end{aligned}$$

The compensation amount Cy and Cz on Y and Z-axes are determined in the same way.

The actual compensation amounts are the calculated compensation amounts multiplied by the compensation magnifications (Parameter No.10809 to 10811).

- Number of compensation points

Up to 15625 compensation points (up to 25 points per axis) can be set. The numbers of compensation points on the individual axes are set for parameters No.10803 to 10805. The ordering of the compensation point numbers in the compensation space is as follows.



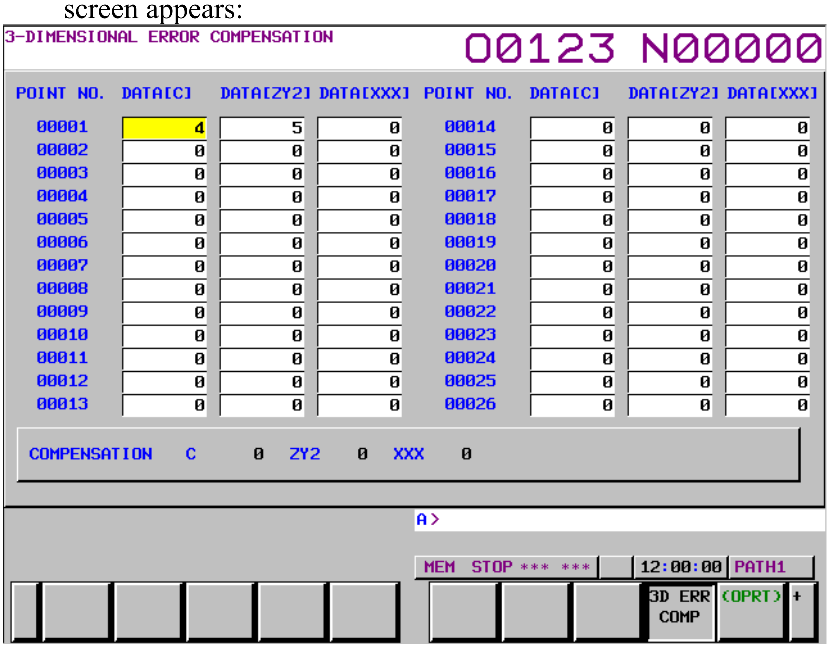
Displaying and setting three-dimensional error compensation data

Procedure

1 Set the following parameters:
- First compensation axis for three-dimensional error compensation : Parmeter (No. 10800)
- Second compensation axis for three-dimensional error compensation : Parmeter (No. 10801)
- Third compensation axis for three-dimensional error compensation : Parmeter (No. 10802)
- Number of compensation points for three-dimensional error compensation (first compensation axis)
  Parmeter (No. 10803)
- Number of compensation points for three-dimensional error compensation (second compensation axis)
  Parmeter (No. 10804)
- Number of compensation points for three-dimensional error compensation (third compensation axis)
  Parmeter (No. 10805)
- Compensation point number of the reference position for three-dimensional error compensation (first compensation axis) : Parmeter (No. 10806)
- Compensation point number of the reference position for three-dimensional error compensation (second compensation axis) : Parmeter (No. 10807)
- Compensation point number of the reference position for three-dimensional error compensation (third compensation axis) : Parmeter (No. 10808)
- Magnification for three-dimensional error compensation (first compensation axis) : Parmeter (No. 10809)
- Magnification for three-dimensional error compensation (second compensation axis) : Parmeter (No. 10810)
- Magnification for three-dimensional error compensation (third compensation axis) : Parmeter (No. 10811)
- Compensation interval for three-dimensional error compensation (first compensation axis)
  Parmeter (No. 10812)
- Compensation interval for three-dimensional error compensation (second compensation axis)
  Parmeter (No. 10813)
- Compensation interval for three-dimensional error compensation (third compensation axis)
  Parmeter (No. 10814)

2 Press function key SYSTEM .

3 Press the continuous menu key several times, then press chapter selection soft key [3D ERR COMP]. The following screen appears:



4 Move the cursor to the position of the compensation point number you want to set using either of the following methods:

- Select the MDI mode.
- Set bit 0 (PWE) of parameter No. 8900 to 1.
- Enter a compensation point number and press soft key [NO. SRH].
- Move the cursor to the compensation point number or compensation axis you want to set by pressing page keys and/or , and cursor keys , , , and .
- Enter compensation data in detection units. The valid data range is from -128 to 127.

12.4.4 Servo Parameters

This subsection describes the initialization of digital servo parameters performed, for example, at the time of field tuning of the machine tool.

Procedure for servo parameter setting

Procedure

1 Turn on the power in the emergency stop state.
2 Set the parameter SVS (No.3111#0) = 1 for displaying the servo setting tuning screen.
3 Turn off the power then turn on the power again.
4 Display the servo parameter setting screen according to the procedure below.

Press the SYSTEM key, ▷, and [SV-PRM] in this order.

The screen below is displayed.

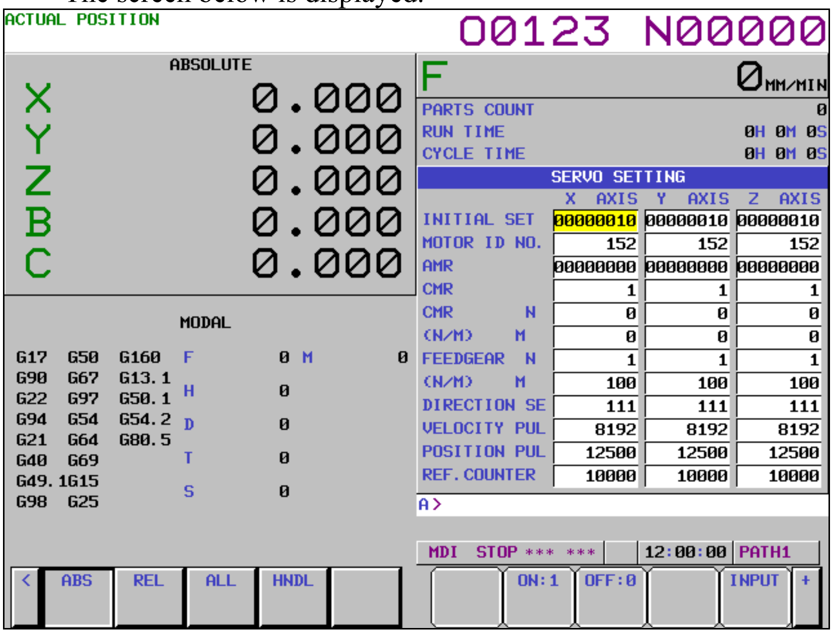


Fig. 12.4.4 (a) Servo parameter setting screen

5 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
6 Key in a desired value then press soft key [INPUT].

12.4.5 Servo Tuning

Data related to servo tuning is displayed and set.

Procedure for servo tuning

Procedure

1 Turn on the power in the emergency stop state.
2 Set the parameter SVS (No.3111#0) = 1 for displaying the servo setting tuning screen.
3 Turn off the power then turn on the power again.
4 Press the SYSTEM key, ▷, and soft key [SV-PRM] in this order.
5 Press soft key [SV.TUN] to select the servo tuning screen.
6 With the cursor keys and page keys, input data necessary for initialization. The screen below is displayed.

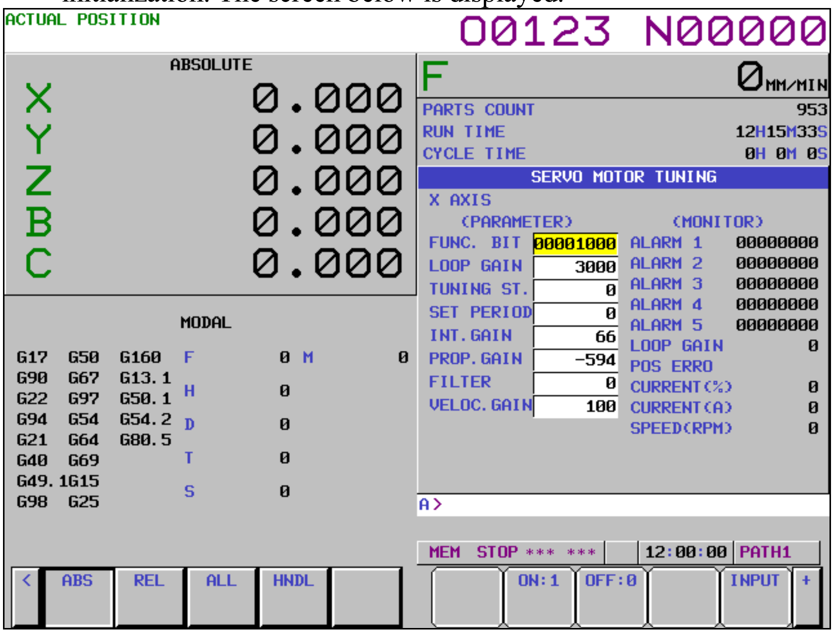


Fig. 12.4.5 (b) Servo tuning screen

7 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
8 Key in a desired value then press soft key [INPUT].

12.4.6 Spindle Setting

Parameters related to spindles are set and displayed. In addition to the parameters, related data can be displayed. Screens for spindle setting, spindle tuning, and spindle monitoring are provided.

Setting spindle parameters

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display the spindle setting and tuning screens.
2 Do the following to display the spindle parameter setting screen:
Press the SYSTEM key, ▷, then [SP.SET].
3 Press the [SP.SET] soft key to select the spindle setting screen.
4 The following screen appears:

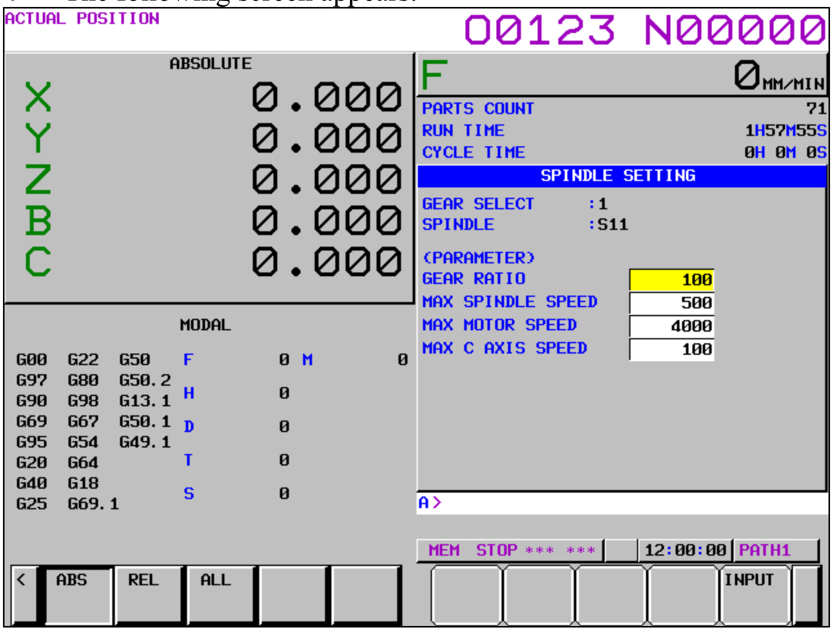


Fig. 12.4.6 (a) Spindle setting screen

5 Move the cursor to the position of the data you want to set or modify by using the page keys and cursor keys.
6 Type a value, then press the [INPUT] soft key.

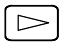
12.4.7 Spindle Tuning

Spindle tuning data is displayed and set.

Setting for spindle tuning

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display the spindle setting and tuning screens.
2 Do the following to display the spindle parameter setting screen:
Press the 

key, , then [SP.SET].
3 Press the [SP.TUN] soft key to select the spindle tuning screen.
4 The following screen appears:

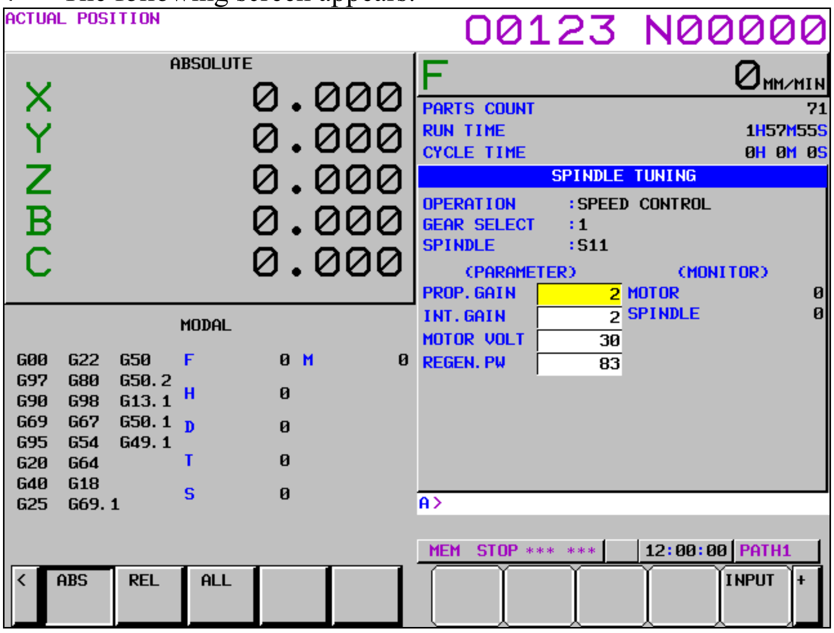


Fig. 12.4.7 (a) Spindle tuning screen

5 Move the cursor to the position of the data you want to set or modify by using the page keys and cursor keys.
6 Type a value, then press the [INPUT] soft key.

12.4.8 Spindle Monitor

Spindle-related data is displayed.

Displaying the spindle monitor

Procedure

1. Set bit 1 (SPS) of parameter No. 3111 to 1 to display the spindle setting and tuning screens.
2. Do the following to display the spindle parameter setting screen:
   Press the SYSTEM key, ▷, then [SP.SET].
3. Press the [SP.MON] soft key to select the spindle monitor screen.
4. The following screen appears:

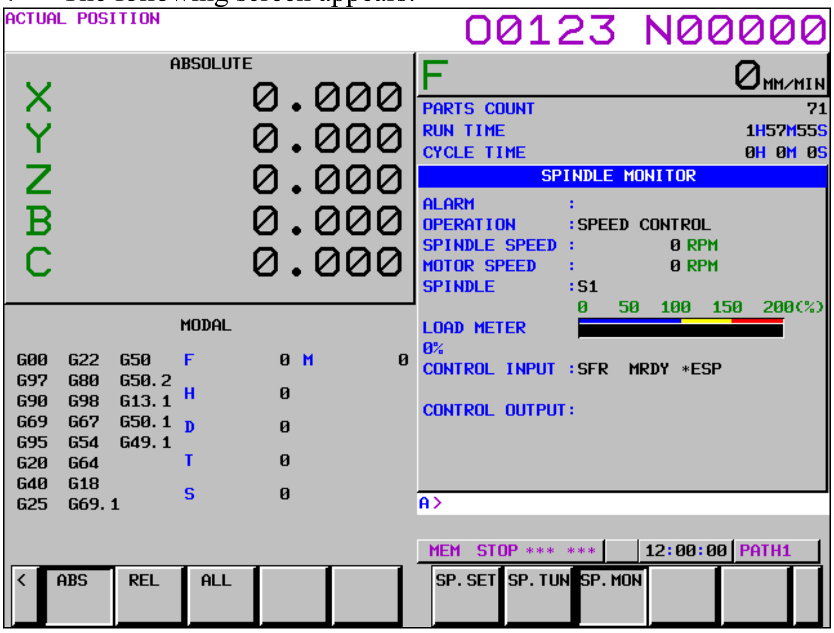


Fig. 12.4.8 (a) Spindle monitor screen

12.4.9 Color Setting Screen

In VGA-compliant screen setting, VGA screen coloring can be performed using the color setting screen.

Displaying the color setting screen

1 Press function key SYSTEM .

2 Press the continuous menu key ▷ several times to display soft key [COLOR].

3 Press soft key [COLOR] to display the color setting screen.

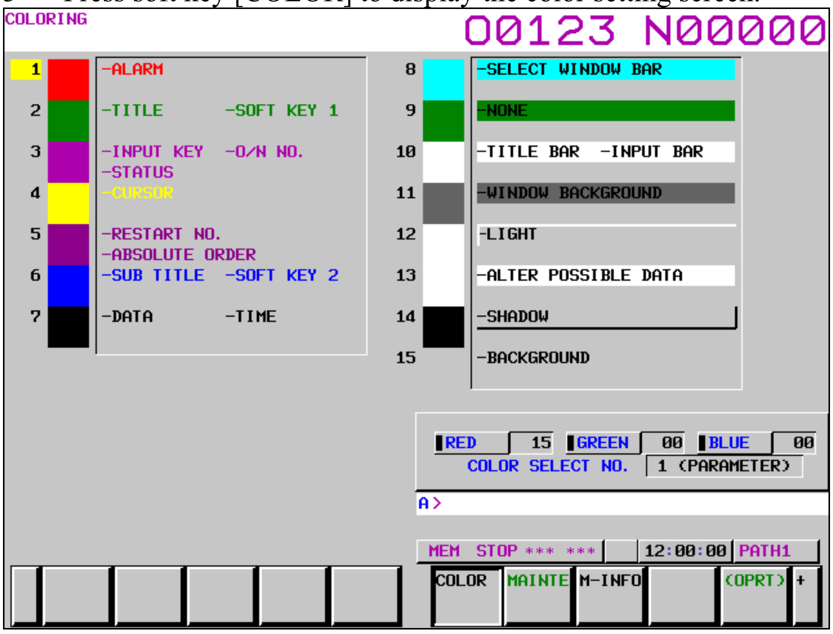
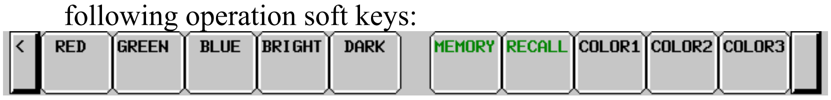


Fig. 12.4.9 (a) Color setting screen

Procedure for operating the color setting screen

- Modifying the color (color palette values)

1 Press soft key [(OPRT)]. The soft key display changes to the following operation soft keys:



2 Move the cursor to a color number whose color palette values are to be modified.
The current color palette value set for each primary color is displayed.

3 Select a primary color whose setting is to be modified, with the corresponding operation soft key [RED], [GREEN], or [BLUE].
More than one primary color can be selected at the same time.
Each time operation soft key [RED], [GREEN], or [BLUE] is pressed, the operation soft key toggles between selection and deselection.

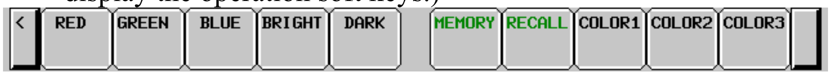
(When operation soft keys [RED], [GREEN], and [BLUE] are not displayed, press the rightmost soft key to display the operation soft keys.)

4 Select operation soft key [BRIGHT] or [DARK] to modify the brightness of the selected prime color(s).

- Storing the color (color palette values)

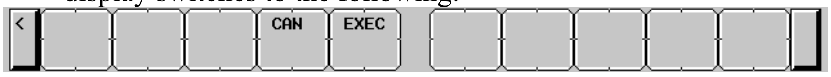
The set color palette values can be stored.

1 Press operation soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area.
(When operation soft keys [COLOR1], [COLOR2], and [COLOR3] are not displayed, press the rightmost soft key to display the operation soft keys.)



COLOR1 Standard color data parameters (Nos. 6581 to 6595)
COLOR2 Parameters (Nos. 10421 to 10435)
COLOR3 Parameters (Nos. 10461 to 10475)

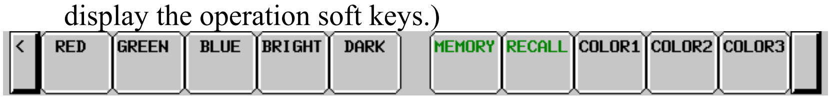
2 Press operation soft key [MEMORY]. The operation soft key display switches to the following:



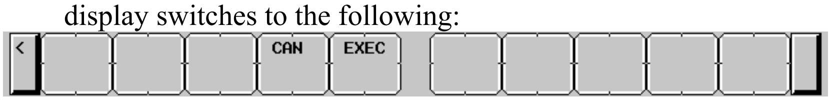
3 Pressing operation soft key [EXEC] stores the current color palette settings in the selected area.
Pressing operation soft key [CAN] or the leftmost key does not store the current color palette settings in the selected area.

- Calling the color (color palette values)

1 Press operation soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area where color palette values are stored.
(When operation soft keys [COLOR1], [COLOR2], and [COLOR3] are not displayed, press the rightmost soft key to display the operation soft keys.)



2 Press operation soft key [RECALL]. The operation soft key display switches to the following:



3 Pressing operation soft key [EXEC] calls the color palette values from the selected area to enable the color to be modified. This operation is invalid if no color palette values are stored.
Pressing operation soft key [CAN] or the leftmost key does not call the color palette values from the selected area.

NOTE

1. Immediately after the power is turned on, the settings of COLOR1 (parameters) are used for display.
   If no values are stored in COLOR1, the color used immediately before the power is turned off is used for display.
2. Do not modify the standard color data parameters directly by MDI key input. When modifying the standard color data, be sure to perform a storage operation on the color setting screen.

12.4.10 Machining Parameter Tuning

In AI contour control, by setting a velocity-emphasized parameter set and precision-emphasized parameter set and setting the precision level matching a machining condition such as rough machining or finish machining on the precision level setting screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on velocity (precision level 1) and emphasis on precision (precision level 10) can be set.
Set the following parameters:

- Acceleration rate of acceleration/deceleration before interpolation
- Acceleration change time (bell-shaped)
- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control
- Allowable acceleration change value for each axis in acceleration change under jerk control in successive linear interpolation operations
- Ratio of the change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation
- Allowable acceleration rate
- Acceleration rate of acceleration/deceleration after interpolation
- Corner speed difference
- Maximum feedrate
- Items that can be set freely (2 items)

For details of each parameter, see the descriptions of AI contour control and jerk control.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a precision level, see the description of the precision level selection screen in Subsection 11.2.14.

Procedure for machining parameter tuning

1 Set the MDI mode.

2 Press function key SYSTEM .

3 Press soft key [M-TUN] to display the machining parameter tuning screen.

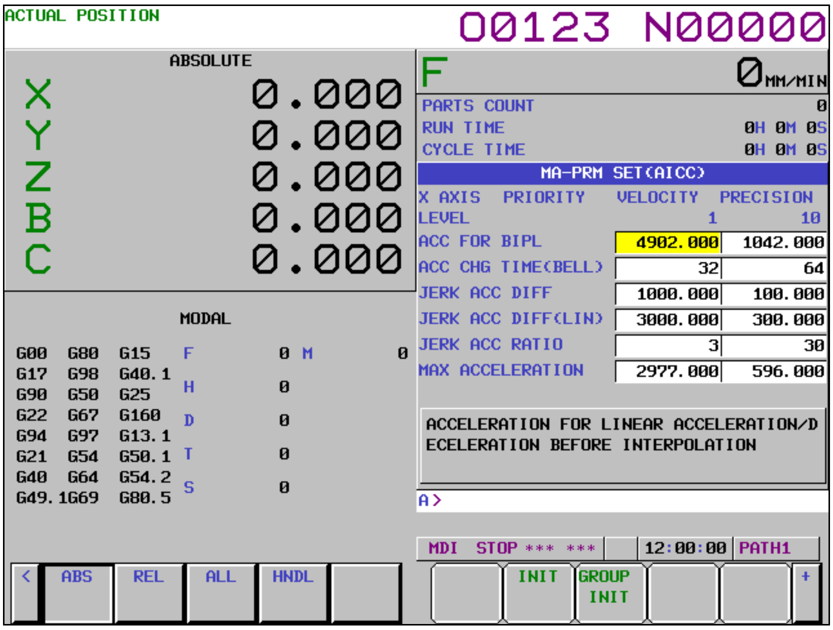


Fig. 12.4.10 (b) Machining parameter tuning screen

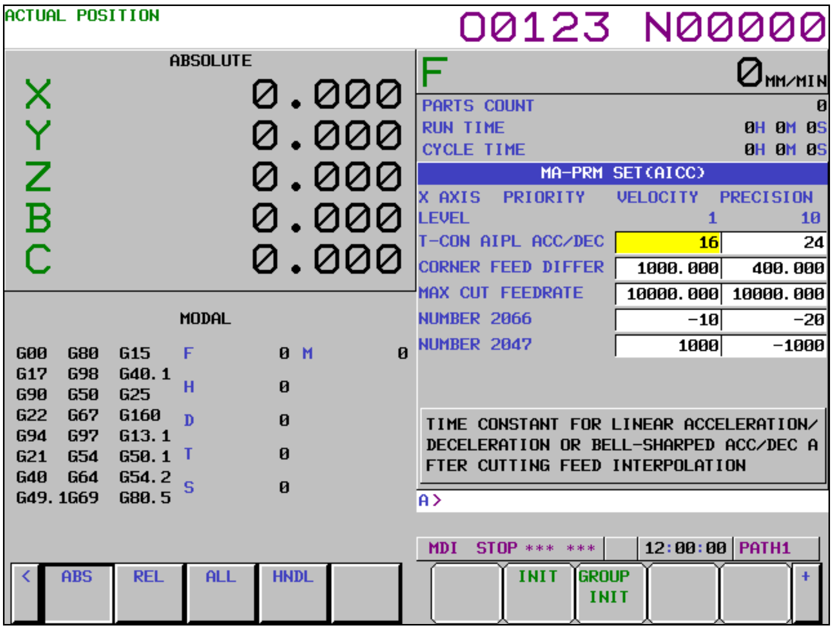


Fig. 12.4.10 (c) Machining parameter tuning screen

4 Move the cursor to the position of a parameter to be set, as follows:

Press page key ↑ PAGE or PAGE ↓, and cursor keys ↑, ↓, ←, and /or → to move the cursor to the parameter.

5 Key in desired data then press the INPUT key on the MDI panel.

6 When data is input, a RMS value is found according to the precision level parameters. (The precision level can be changed on the precision level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.

7 Repeat steps 2 and 3 until all machining parameters are set.
8 In addition to the setting method described above, a parameter setting method using soft keys is available. Pressing soft key [INIT] displays the standard value (recommended by FANUC) of the item selected by the cursor in the key input buffer. Pressing soft key [EXEC] initializes the item to the standard value. Pressing soft key [GROUP INIT] initializes all items of a group (emphasis on velocity or emphasis on precision) selected by the cursor to the standard values.

The table below indicates the initial settings.

Table 12.4.10 (a) Initial settings

Setting item	AI contour control		Unit
	Emphasis on velocity (LV1)	Emphasis on precision (LV10)	
Acceleration rate of acceleration/deceleration before interpolation <ACC FOR BIPL>	4902.000	1042.000	mm/sec^2
Acceleration change time (bell-shaped) <ACC CHG TIME(BELL)>	32	64	msec
Allowable acceleration change value <JERK ACC DIFF>	0	0	mm/sec^2
Allowable acceleration change value in successive linear interpolation operations <JERK ACC DIFF(LIN)>	0	0	mm/sec^2
Ratio of the change time of the jerk control <JERK ACC RATIO>	0	0	%
Allowable acceleration rate <MAX AACELERATION>	2977.000	596.000	mm/sec^2
Time constant for acceleration/deceleration after interpolation <T-CON AIPL ACC/DEC>	24	24	msec
Corner speed difference <CORNER FEED DIFFER>	1000	400	mm/min
Maximum cutting speed <MAX CUT FEEDRATE>	10000	10000	mm/min

Explanation

- Look-ahead acceleration/deceleration before interpolation

Set an acceleration rate for a linear portion in look-ahead acceleration/deceleration before interpolation.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13610 (velocity-emphasized parameter)
Parameter No. 13611 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1660: Maximum allowable acceleration rate for each axis in acceleration/deceleration before interpolation

- Acceleration change time (bell-shaped)

Set a time constant for a bell-shaped portion in acceleration/ deceleration before look-ahead interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13612 (velocity-emphasized parameter)
Parameter No. 13613 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1772: Time constant for bell-shaped look-ahead acceleration/deceleration before interpolation of constant acceleration time type

> ⚠ **CAUTION**
> A set time constant is applied to all axes. So, a modification made to this item changes the settings for all axes.

- Allowable acceleration change value in velocity control based on acceleration change under jerk control

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13614 (velocity-emphasized parameter)
Parameter No. 13615 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1788: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control

> ⚠ **CAUTION**
> This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13616 (velocity-emphasized parameter)
Parameter No. 13617 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1789: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

> ⚠ **CAUTION**
>
> 1 For an axis with 0 set in this parameter, the parameters (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615) are valid.
>
> 2 For an axis with 0 set in the parameter (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615), velocity control based on acceleration change is disabled, so that this parameter has no effect.
>
> 3 This setting item is displayed only when the jerk control function is enabled.

- Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

Unit of data: %

Set the ratio (in %) of the change time of jerk control to the change time of acceleration in smooth bell-shaped acceleration/deceleration before interpolation.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13618 (velocity-emphasized parameter)
Parameter No. 13619 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1790: Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

> ⚠ **CAUTION**
> This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration rate

Set an allowable acceleration rate in acceleration-based speed determination.
Unit of data: mm/sec^2, $inch/sec^2$, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13620 (velocity-emphasized parameter)
Parameter No. 13621 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1735: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in circular interpolation
Parameter No. 1737: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in AI contour control

> ⚠ **CAUTION**
> When bit 0 (MCR) of parameter No. 13600 is set to 1, the deceleration function based on acceleration in circular interpolation is not set.

- Time constant for acceleration/deceleration after interpolation

Set a time constant for acceleration/deceleration after interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13622 (velocity-emphasized parameter)
Parameter No. 13623 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1769: Time constant for acceleration/deceleration after cutting feed interpolation

- Corner speed difference

Set an allowable corner speed difference used for speed determination.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13624 (velocity-emphasized parameter)
Parameter No. 13625 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1783: Allowable speed difference for each axis in automatic corner deceleration based on speed difference

- Maximum cutting speed

Set a maximum cutting speed for each axis.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13626 (velocity-emphasized parameter)
Parameter No. 13627 (precision-emphasized parameter)

Moreover, the following parameter is also set from the precision level:
Parameter No. 1432: Maximum cutting feedrate for each axis in the AI contour control mode

- Arbitrary items

Two arbitrary parameters can be registered. Each item can correspond to a CNC parameter or servo parameter. A parameter number corresponding to each item is to be specified with parameters.

As indicated below, set the parameters for corresponding parameter numbers, velocity-emphasized parameters (precision level 1), and precision-emphasized parameters (precision level 10).

Table 12.4.10 (a) Parameters related to arbitrary items

	Corresponding parameter number	Setting of velocity-emphasized (precision level 1) value	Setting of precision-emphasized (precision level 10) value
Arbitrary item 1	No.13628	No.13630	No.13632
Arbitrary item 2	No.13629	No.13631	No.13633

- Display
  Tuning target parameter numbers are displayed.

> ⚠ **CAUTION**
> As arbitrary items, the numbers of the following parameters cannot be specified:
> - Bit parameter
> - Spindle parameters (Parameter Nos. 4000 to 4799)
> - Real-type parameter
> - Power-off parameter
> - Nonexistent parameter

12.4.11 Displaying Memory Data

The contents of the CNC memory can be displayed starting at a specified address.

Displaying memory data

Procedure

1 Set bit 0 (MEM) of parameter No. 8950 to 1 to display the memory contents display screen.

2 Press function key SYSTEM .

3 Press the continuous menu key , then press chapter selection soft key [MEMORY].
The screen below is displayed.

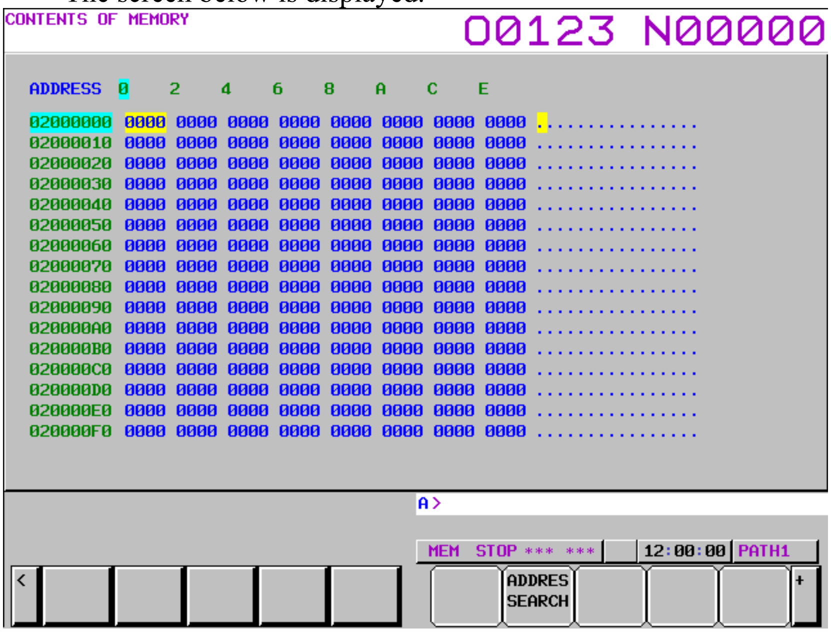


Fig. 12.4.11 (a) Memory contents display screen

4 Key in a desired address (hexadecimal) then press the [ADDRES SEARCH] key. Starting at the specified address, 256-byte data is displayed.
(Example: When you input 100000 then press [ADDRES SEARCH], data is displayed starting at 100000H.)

5 Display data can be switched by using page keys PAGE and PAGE and cursor keys , , , and/or .

6 By pressing soft key [BYTE], [WORD], [LONG], or [DOUBLE], select a data type to be displayed.

Explanation

A memory data display format can be selected from the following four options:

Byte display (1 byte in hexadecimal)
Word display (2 bytes in hexadecimal)
Long display (4 bytes in hexadecimal)
Double display (8 bytes in decimal: Double precision floating-point display)

One screen displays 256-byte memory data.

NOTE
1 When an address is input, "H" for representing "hexadecimal" need not be specified at the end of the address. If H is added, a warning is displayed to indicate that the format is invalid.
2 When word display is selected as the display format, an input address is rounded to a multiple of 2 bytes. When long display or double display is selected, an input address is rounded to a multiple of 4 bytes.

⚠ **WARNING**
1 If a memory address that must not be accessed in address search is input, a system alarm is issued. When making an address search, check that the address is accessible and that the address is input correctly.
2 This function is designed for maintenance, and must not be used by general users.

12.4.12 Parameter Tuning Screen

The parameter tuning screen is a screen for parameter setting and tuning designed to achieve the following:

1. The minimum required parameters that must be set when the machine is started up are collectively displayed to facilitate start-up of the machine.
2. The servo tuning screen, spindle tuning screen, and machining parameter tuning screen are displayed for smooth tuning.

The parameter tuning screen consists of a menu screen and several setting screens.

12.4.12.1 Displaying the menu screen and selecting a menu item

The parameter tuning menu screen displays the following items:
[START UP]

- SYSTEM SETTING
- AXIS SETTING
- FSSB (AMP)
- FSSB (AXIS)
- SERVO SETTING
- SPINDLE SETTING
- MISCELLANY

[TUNING]

- SERVO TUNING
- SPINDLE TUNING
- AICC TUNING

On the parameter tuning menu screen, one of the displayed items can be selected to display the corresponding screen. From each setting screen, you can return to the menu screen by performing a soft key operation.

> **NOTE**
> 1 Some items may not be displayed, depending on the system configuration.
> 2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

Displaying the menu screen and selecting a setting screen

Procedure

1 Set the MDI mode.
2 Switch the setting of "PARAMETER WRITE" to "ENABLED". For details, see the procedure for "PARAMETER WRITE" in Subsection III-12.4.1.
3 Press function key SYSTEM .
4 Press the rightmost soft key ▷ (continuous menu key) several times.
5 Press soft key [PRMTUN] to display the parameter tuning menu screen.

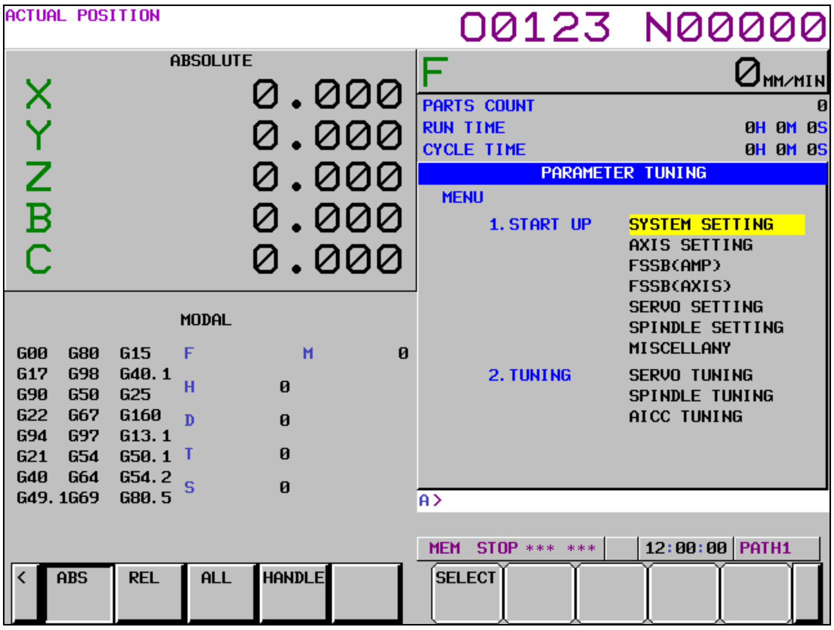


Fig. 12.4.12 (a) Parameter tuning menu screen

7 Move the cursor to a desired item by pressing cursor key ↑ or ↓.
8 Press soft key [SELECT]. The screen display switches to the selected screen.

Returning to the menu screen

Procedure

1 Press soft key [SELECT] on the parameter tuning menu screen described in Subsection III-12.4.13.1. The screen and soft keys shown below are displayed. (The screen below is displayed when "AXIS SETTING" is selected.)

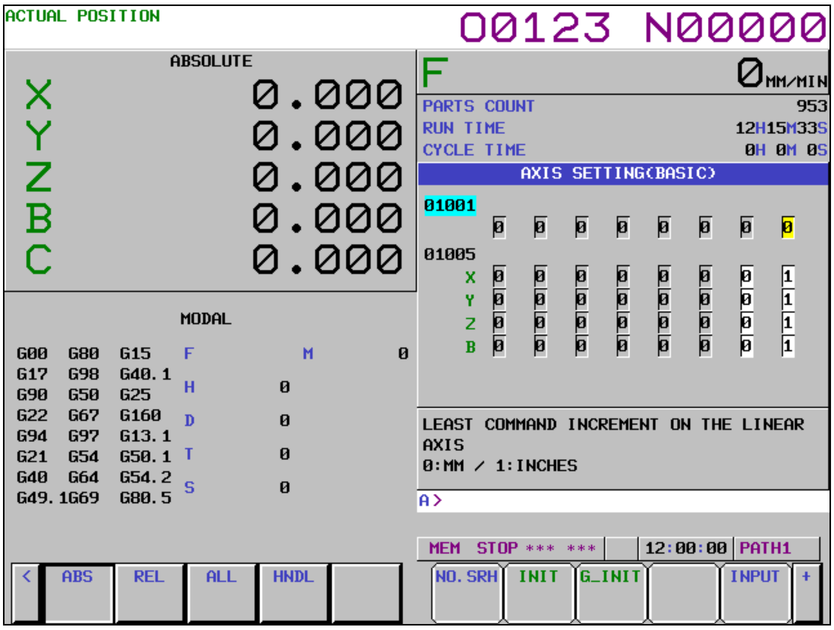


Fig. 12.4.12 (b) Axis setting screen

2 Press the rightmost soft key (continuous menu key) several times.

3 Press soft key [MENU].
The screen display returns to the parameter tuning menu screen.

4 Upon completion of parameter setting, switch the setting of "PARAMETER WRITE" to "DISABLED".

> **NOTE**
> Some setting screens can also be displayed by a chapter selection soft key. If a screen is selected using a chapter selection soft key, however, you cannot return to the parameter tuning menu screen.

Explanation

- Items displayed with [START UP]

The items of [START UP] indicate the screens for setting the minimum required parameters for starting up the machine.

Table 12.4.12 (a) Items displayed with [START UP]

Display item	Description
SYSTEM SETTING	Screen for setting CNC parameters related to the entire system configuration
AXIS SETTING	Screen for setting CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration
FSSB (AMP)	FSSB amplifier setting screen
FSSB (AXIS)	FSSB axis setting screen
SERVO SETTING	Servo setting screen
SPINDLE SETTING	Screen for setting spindle-related parameters
MISCELLANY	Screen for setting parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens

NOTE
Some items may not be displayed, depending on the system configuration.

- Items displayed with [TUNING]

The items of [TUNING] indicate the screens for servo, spindle, and high-speed high-precision machining tuning.

Fig. 12.4.12 (c) Items displayed with [TUNING]

Display item	Description
SERVO TUNING	Servo tuning screen
SPINDLE TUNING	Spindle tuning screen
AICC TUNING	Machining parameter tuning screen

NOTE
1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

12.4.12.2 Parameter tuning screen (system setting)

This screen enables the parameters related to the entire system configuration to be displayed and modified. The parameters can be initialized to the standard values (recommended by FANUC).

Display and setting

Procedure

1 Move the cursor to [SYSTEM SETTING] by pressing cursor key ↑ or ↓ on the parameter tuning menu screen described in Subsection III-12.4.13.1.

2 Press soft key [SELECT]. The screen display switches to the screen and soft keys shown below.

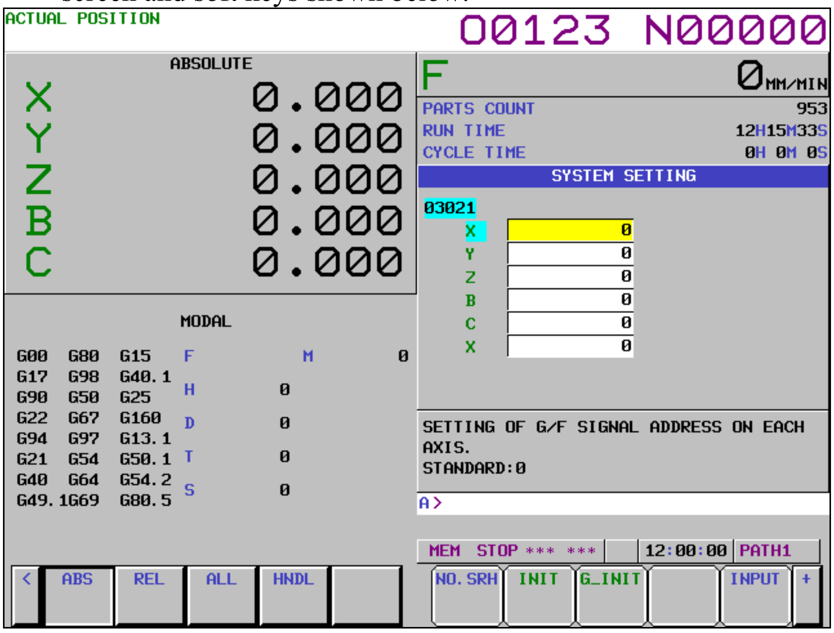


Fig. 12.4.12 (d) Parameter tuning screen (system setting)

3 Move the cursor to a parameter number to be set or displayed, according to one of the methods below.

- Input the desired parameter number, then press soft key [NO.SRH].
- Move the cursor to the desired parameter number by pressing page key PAGE ↑ or PAGE ↓, and cursor keys ↑, ↓, ←, and/or →.

A brief description of the parameter where the cursor is placed is provided at the bottom of the screen. However, no description is provided when the cursor is placed on multiple bits for bit parameters.

4 Input desired data then press the INPUT key on the MDI panel to set the parameter.

5 Press soft key [INIT]. The standard value (recommended by FANUC) for the item selected by the cursor is displayed in the key input buffer. Pressing soft key [EXEC] in this state initializes the item to the standard value.

6 Press soft key [G_INIT]. A message asking whether to set the group standard values is displayed on the screen. Pressing soft key [EXEC] in this state inputs all of the standard values of the group.

NOTE

1 If the cursor is placed on a parameter that has no standard value assigned, no standard value is input even when [INIT] is pressed.

2 When the cursor is placed on multiple bits for bit parameters, the multiple bits can be input simultaneously. When [INIT] is pressed in this state, the key input buffer displays the standard values for the bits where the cursor is placed. If a bit has no standard value assigned, "*" is displayed for the bit, and no value is input for the bit.

3 When [G_INIT] is pressed, those parameters that have no standard values assigned are not initialized.

12.4.12.3 Parameter tuning screen (axis setting)

This screen enables the CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration to be displayed and set. The parameters displayed can be divided into four groups:

(Basic) group :

The parameters related to basic settings are displayed.

(Coordinate) group :

The parameters related to coordinates are displayed.

(Feedrate) group :

The parameters related to feedrate are displayed.

(Acceleration/deceleration) group:

The parameters related to acceleration/deceleration are displayed.

The parameters can be initialized to the standard values (recommended by FANUC). For the display and setting procedure, see the description of the parameter tuning screen (axis setting) in Subsection III-12.4.13.2.

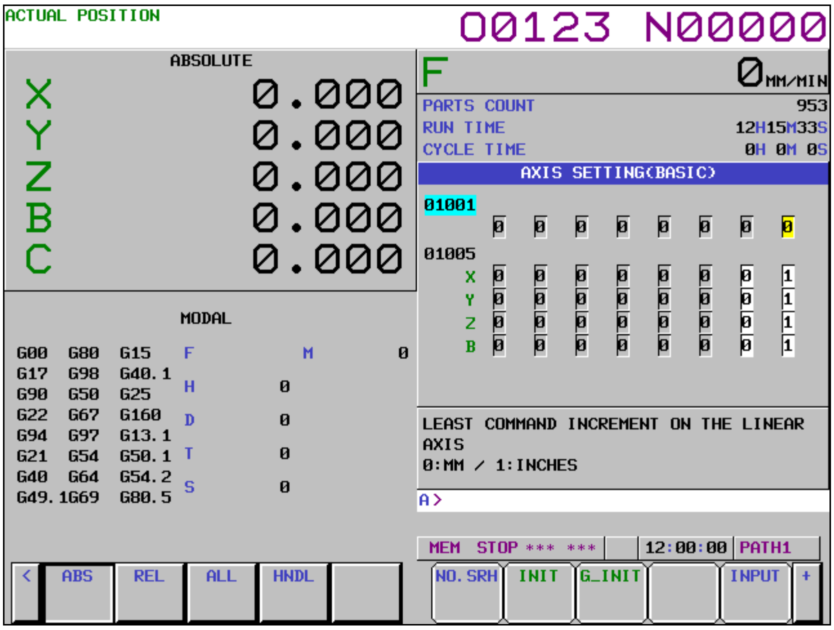


Fig. 12.4.12 (e) Parameter tuning screen (axis setting)

12.4.12.4 Displaying and setting the FSSB amplifier setting screen

From the parameter tuning screen, the FSSB amplifier setting screen can be displayed. For details of the FSSB amplifier setting screen, see the description of the FSSB amplifier setting screen in Subsection 1.4.4 in the CONNECTION MANUAL (FUNCTION) (B-63943EN-1).

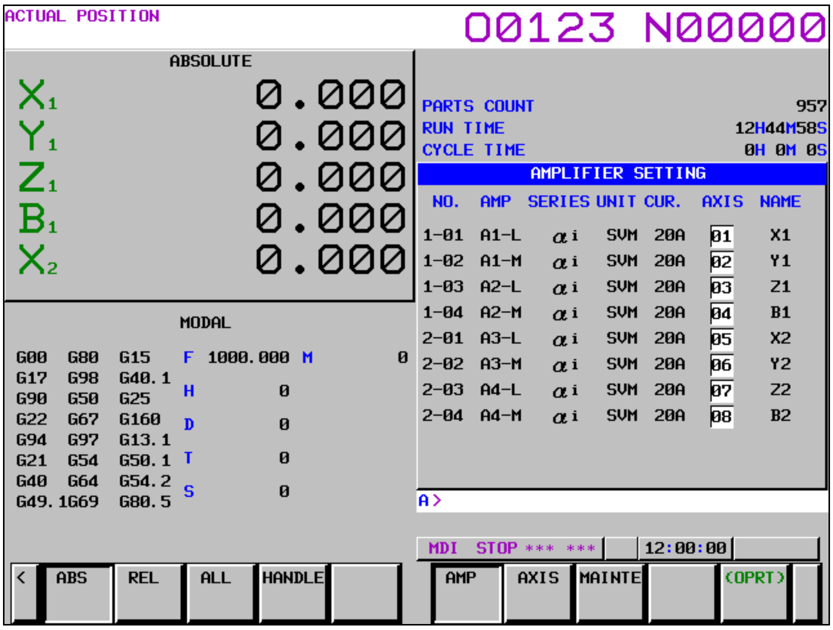


Fig. 12.4.12 (f) FSSB amplifier setting screen

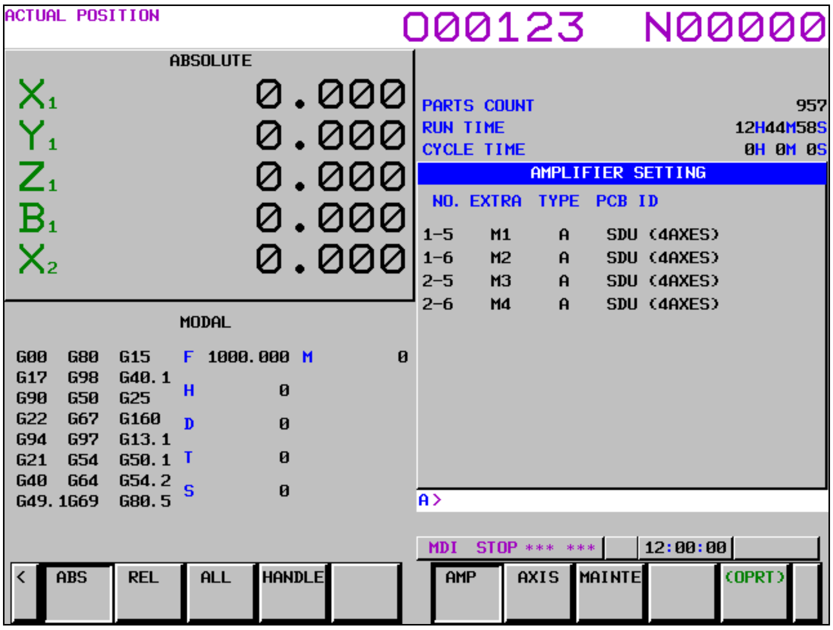


Fig. 12.4.12 (g) FSSB amplifier setting screen 2

12.4.12.5 Displaying and setting the FSSB axis setting screen

From the parameter tuning screen, the FSSB axis setting screen can be displayed. For details of the FSSB axis setting screen, see the description of the FSSB axis setting screen Subsection 1.4.4 in the CONNECTION MANUAL (FUNCTION) (B-63943EN-1).

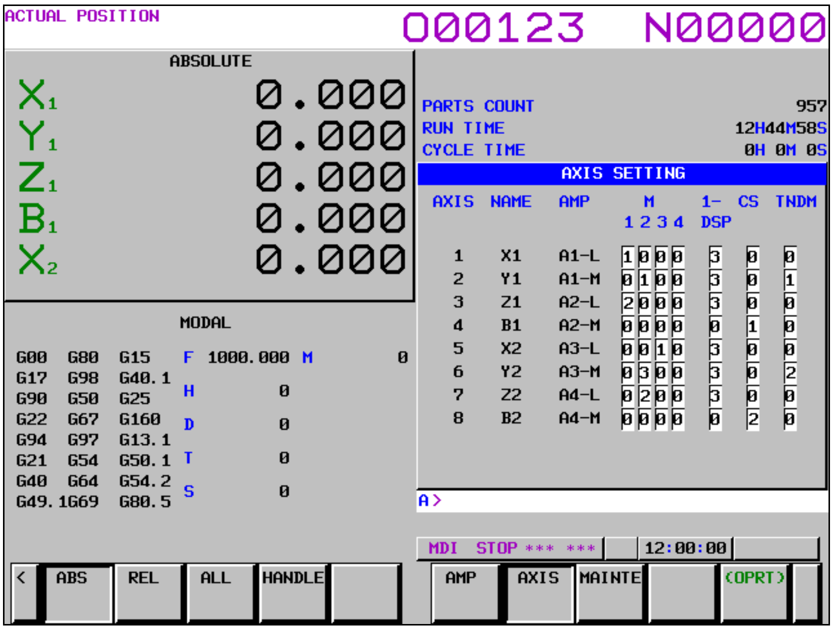


Fig. 12.4.12 (h) FSSB axis setting screen

12.4.12.6 Displaying and setting the servo setting screen

From the parameter tuning screen, the servo setting screen can be displayed. For details of the servo setting screen, see the description of the servo setting screen in Subsection III-12.4.4.

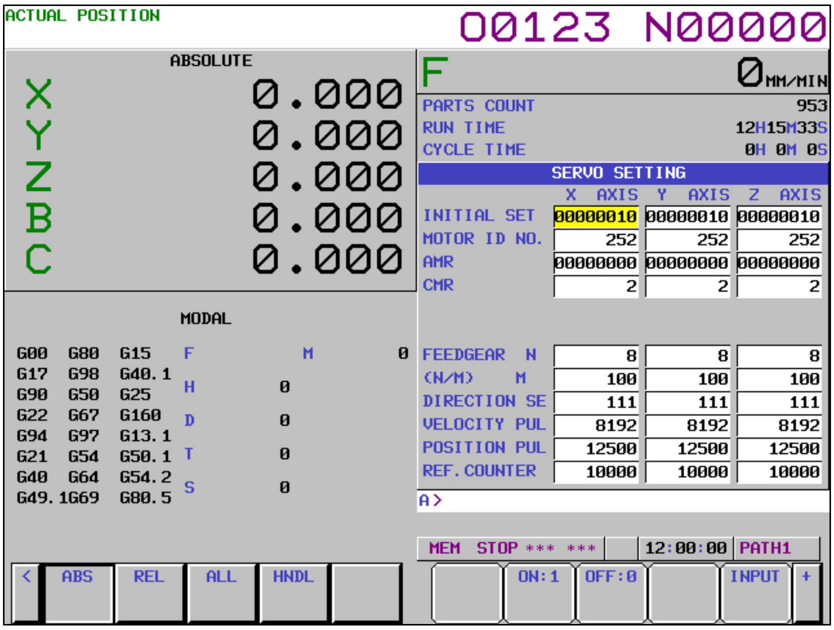


Fig. 12.4.12 (i) Servo setting screen

12.4.12.7 Parameter tuning screen (spindle setting)

The spindle-related parameters can be displayed and modified.
For the display and setting procedure, see the description of the parameter tuning screen (system setting) in Subsection III-12.4.13.2.

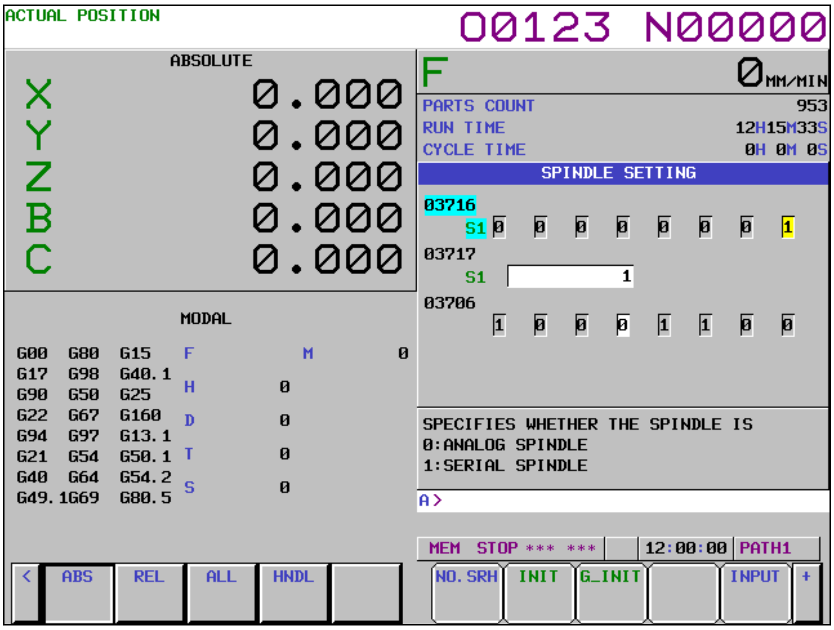


Fig. 12.4.12 (j) Parameter tuning screen (spindle setting)

12.4.12.8 Parameter tuning screen (miscellaneous settings)

The parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens can be displayed and modified. Moreover, the parameters can be initialized to the standard values (recommended by FANUC).
For the display and setting procedure, see the description of the parameter tuning screen (system setting) in Subsection III-12.4.13.2.

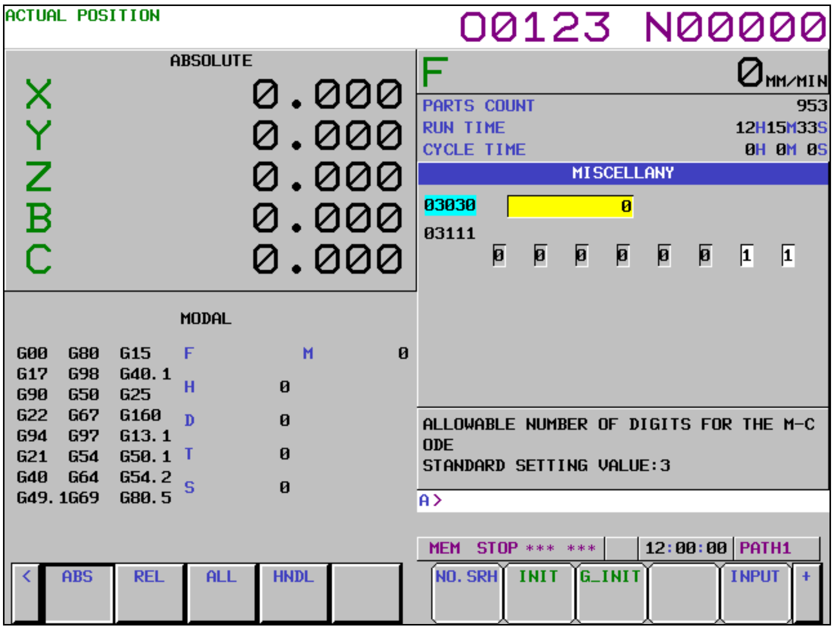


Fig. 12.4.12 (k) Parameter tuning screen (system setting)

12.4.12.9 Displaying and setting the servo tuning screen

From the parameter tuning screen, the servo tuning screen can be displayed. For details of the servo tuning screen, see the description of the servo tuning screen in Subsection III-12.4.5.

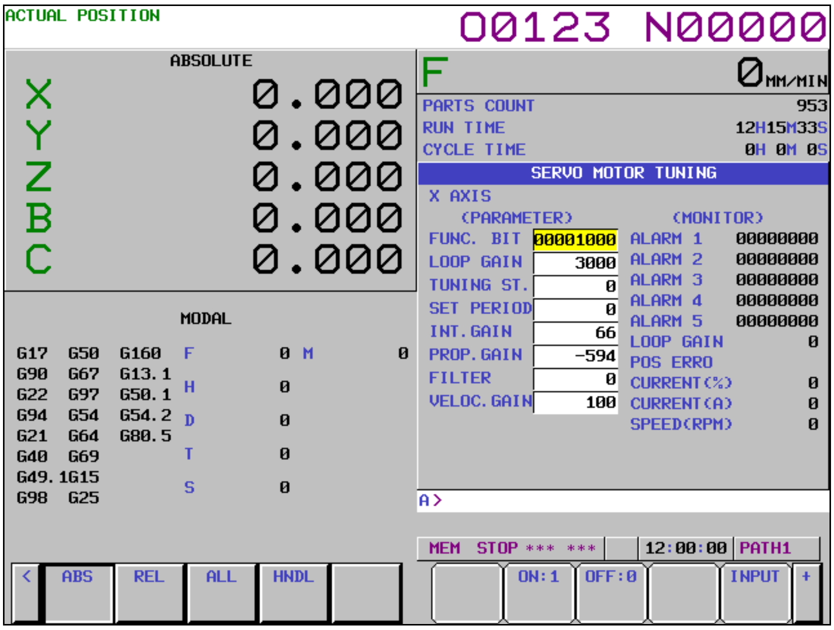


Fig. 12.4.12 (l) Servo tuning screen

12.4.12.10 Displaying and setting the spindle tuning screen

From the parameter tuning screen, the spindle tuning screen can be displayed. For details of the spindle tuning screen, see the description of the spindle tuning screen in Subsection III-12.4.7.

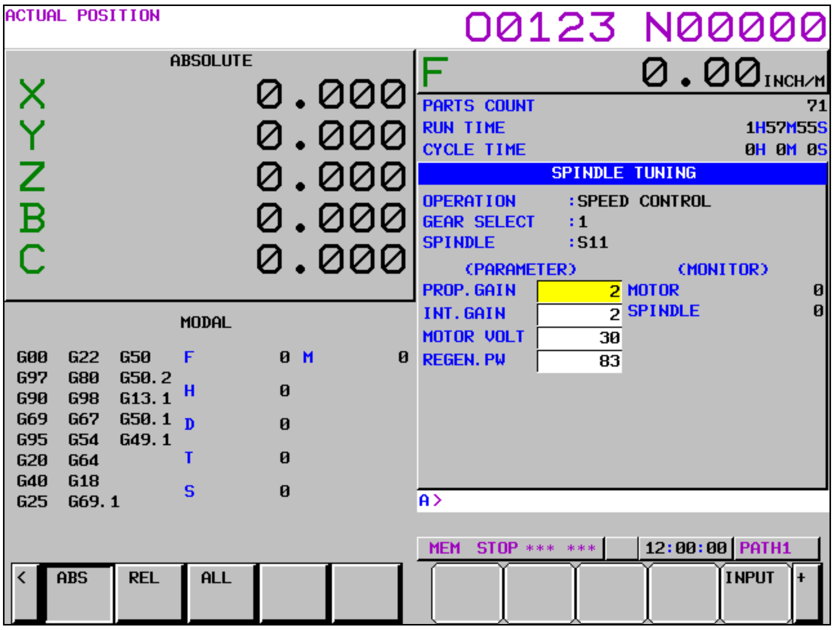


Fig. 12.4.12 (m) Spindle tuning screen

12.4.12.11 Displaying and setting the machining parameter tuning screen

From the parameter tuning screen, the machining parameter tuning screen can be displayed. For details of the machining parameter tuning screen, see the description of the machining parameter tuning screen in Subsection III-12.4.10.

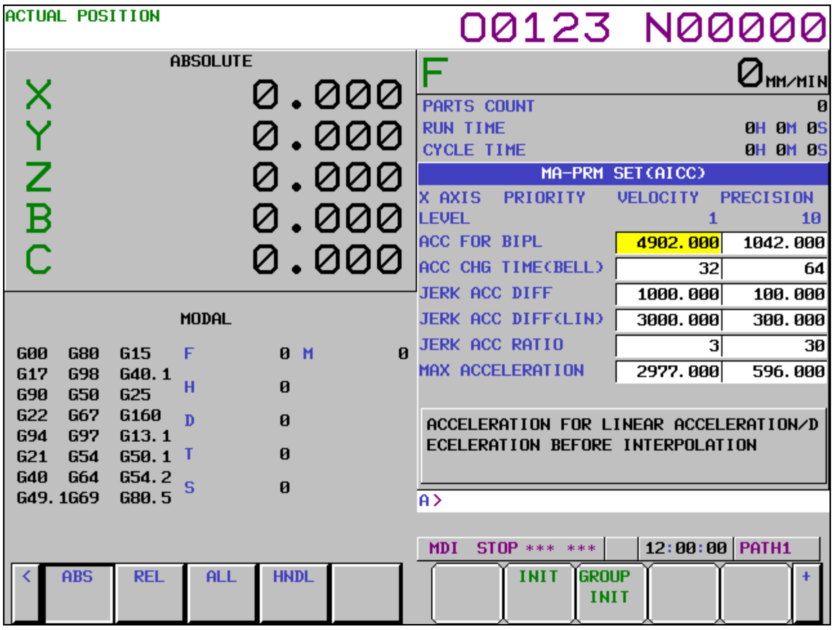


Fig. 12.4.12 (n) Machining parameter tuning screen

Explanation

- Parameters displayed for parameter tuning

Table 12.4.12 (b) Parameters displayed for parameter tuning (1)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SYSTEM SETTING	System setting	981		Sets the path of each axis.	
		982		Sets the path of each spindle.	
		983		Sets the T series/M series of each path. 0:T series/1:M series	
		984#0	LCP	Sets the attribute of each path. 0:Normal/1:Loader control	*1
		3021		Sets the G/F signal address of each axis.	*2
		3022		Sets the G/F signal address of each spindle.	*3
		3006#0	GDC	The deceleration signal for reference position return is: 0:X signal/1:G signal	1
		3008#2	XSGx	The signal address allocated to address X is: 0:Fixed/1:Set by the parameter	1
		3013		Address allocated to the deceleration signal for reference position return	*4
		3014		Bit position allocated to the deceleration signal for reference position return	*5

*1 : The value 1 is set for the paths as many as the number of loader paths starting from the greatest path number. For path 1, the value 0 is set at all times.
Example) When the number of loader paths is 3 in a 10-path system:
The value 1 is set for paths 8 to 10. The value 0 is set for others.

*2 : When intra-path axis number ≤ 8, (path number - 1)*10+(intra-path axis number - 1)
When intra-path axis number ≥ 9, no standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 10,11,12 for axes of path 2

*3 : When intra-path spindle number ≤ 4, (path number - 1)*10+(intra-path spindle number - 1)
When intra-path spindle number ≥ 5, no standard value is available.
Example) When path 1 has 5 spindles, and path 2 has 1 spindle:
0,1,...,4,(none) for spindles of path 1, 10 for spindle of path 2

*4 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
Axes of path 1: 9
Axes of path 2: 7
Axes of path 3: 10
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
9,9,9,9,9,9,9,9,(none) for axes of path 1, 7,7,7 for axes of path 2

*5 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
(Intra-path axis number - 1)
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 0,1,2 for axes of path 2

Table 12.4.12 (c) Parameters displayed for parameter tuning (2)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SPINDLE SETTING	Spindle setting	3716#0	A/S	Sets the type of spindle motor: 0:Analaog/1:Serial.	
		3717		Sets a motor number to be assigned to each spindle.	
		3706#4	GTT	Specifies a spindle gear selection method. 0:M type/1:T type	
		3718		Sets a suffix to be added to spindle speed display on a screen such as the position display screen.	
		3735		Sets the minimum clamp speed of the spindle motor.	
		3736		Sets the maximum clamp speed of the spindle motor.	
		3741		Sets the maximum spindle speed for gear 1.	
		3742		Sets the maximum spindle speed for gear 2.	
		3743		Sets the maximum spindle speed for gear 3.	
		3744		Sets the maximum spindle speed for gear 4.	
		3772		Sets the maximum speed of the spindle. When 0 is set, the speed is not clamped.	
		4133		Sets the motor model code of the serial spindle. (This setting is not required for an analog spindle.)	
		4019#7	***	The parameters for the serial spindle are: 0:Not automatically set/1:Automatically set (This setting is not required for an analog spindle.)	

Table 12.4.12 (d) Parameters displayed for parameter tuning (3)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Basic	1001#0	INM	Least command increment on linear axes: 0:Metric (millimeter machine) 1:Inch (inch machine)	
		1005#0	ZRNx	When an automatic operation (other than G28) is executed before reference position return: 0:Alarm is issued (No.224) 1:No alarm is issued.	0
		1005#1	DLZx	Reference position return without dogs: 0:Disabled 1:Enabled	
		1006#0	ROTx	Setting of linear axes or rotary axis: 0:Linear axis 1:Rotation axis	
		1006#3	DIAx	Setting of the amount of travel: 0:Radius specification 1: Diameter specification	
		1006#5	ZMIx	Reference position return direction: 0:Plus direction 1:Minus direction	
		1008#0	ROAx	Rotation axis roll-over function: 0:Disabled 1:Enabled	1
		1008#2	RRLx	With the amount of travel per revolution, relative coordinates are: 0:Not rounded 1:Rounded	1
		1013#0	ISAx	Sets the least input increment and least command increment: 0:IS-B 1:IS-A	
		1013#1	ISCx	Sets the least input increment and least command increment: 0:IS-B 1:IS-C	
		1013#2	ISDx	Sets the least input increment and least command increment: 0:IS-B 1:IS-D	
		1013#3	ISEx	Sets the least input increment and least command increment: 0:IS-B 1:IS-E	
		1020		Program name	*1
		1022		Sets each axis in the basic coordinate system.	*2
		1023		Servo axis number	
		1815#1	OPTx	A separate pulse coder is: 0:Not used 1:Used	
		1815#4	APZx	The correspondence between machine positions and absolute-position detector positions is: 0:Not established 1:Established	
		1815#5	APCx	The position detector used is: 0:Other than an absolute-position detector 1:Absolute-position detector	
		1825		Servo loop gain	
		1826		Effective area	
		1828		Positional deviation limit during travel	
		1829		Positional deviation limit during stop	500

*1 : For the M series: 88(X),89(Y),90(Z) sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 88(X),90(Z) (no standard value for the third axis and up)

*2 : For the M series: 1,2,3 sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 1,3 (no standard value for the third axis and up)

Table 12.4.12 (e) Parameters displayed for parameter tuning (4)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Coordinate	1240		Machine coordinate of the first reference position	
		1241		Machine coordinate of the second reference position	
		1260		Amount of travel per revolution of a rotary axis	360.000
		1320		Coordinate of the boundary of stored stroke check 1 in the positive direction	
		1321		Coordinate of the boundary of stored stroke check 1 in the negative direction	
	Feedrate	1401#6	RDR	For a rapid traverse command, dry run is: 0:Disabled 1:Enabled	0
		1410		Dry run feedrate	
		1420		Rapid traverse rate	
		1421		Rapid traverse override F0 rate	
		1423		Jog feedrate	
		1424		Manual rapid traverse rate	
		1425		FL feedrate for reference position return	
		1428		Reference position return feedrate	
		1430		Maximum cutting feedrate	
	Acceleration/ deceleration	1610#0	CTLx	Acceleration/deceleration for cutting feed is: 0:Exponential acceleration/deceleration 1:Linear acceleration/deceleration after interpolation	
		1610#4	JGLx	Acceleration/deceleration for jog feed is: 0:Exponential acceleration/deceleration 1:Same as acceleration/deceleration for cutting feed (The settings of bit 1 (CTBx) and bit 0 (CTLx) of parameter No. 1610 are followed.)	
		1620		Time constant for linear acceleration/deceleration for rapid traverse	
		1622		Time constant for acceleration/deceleration for cutting feed	
		1623		FL feedrate for acceleration/deceleration after interpolation for cutting feed	
		1624		Time constant for acceleration/deceleration for jog feed	
		1625		FL feedrate for exponential acceleration/deceleration for jog feed	
MISCELLANY	MISC	3030		Allowable number of digits of an M code	3
		3111#0	SVS	The servo setting screen is: 0:Not displayed 1:Displayed	1
		3111#1	SPS	The spindle setting screen is: 0:Not displayed 1:Displayed	1

12.5 SCREENS DISPLAYED BY FUNCTION KEY MESSAGE

By pressing the function key MESSAGE , data such as alarms, and alarm history data can be displayed.
For information relating to alarm display, see Section III.7.1. For information relating to alarm history display, see Section III.7.2.

12.6 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION

The program number, sequence number, and current CNC status are always displayed on the screen except when the power is turned on, a system alarm occurs, or the PMC screen is displayed.
If data setting or the input/output operation is incorrect, the CNC does not accept the operation and displays a warning message.
This section describes the display of the program number, sequence number, and status, and warning messages displayed for incorrect data setting or input/output operation.

12.6.1 Displaying the Program Number and Sequence Number

The program number and sequence number are displayed at the top right on the screen as shown below.

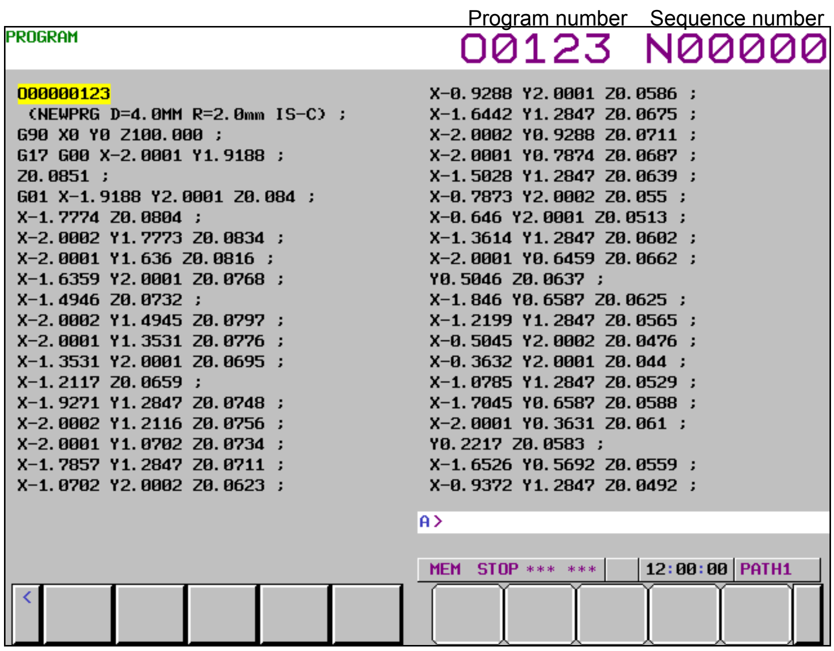


Fig. 12.6.1 (a)

The program number and sequence number displayed depend on the screen and are given below:

- **On the program screen in the EDIT mode on Background edit screen :**
  The program No. being edited and the sequence number just prior to the cursor are indicated.
- **Other than above screens :**
  The program No. and the sequence No. executed last are indicated.

- **Immediately after program number search or sequence number search :**
  Immediately after the program No. search and sequence No. search, the program No. and the sequence No. searched are indicated.

12.6.2 Displaying the Status and Warning for Data Setting or Input/Output Operation

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the screen allowing the operator to readily understand the operation condition of the system. If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed on the next to last line of the screen. This prevents invalid data setting and input/output errors.

Explanation

- Description of each display

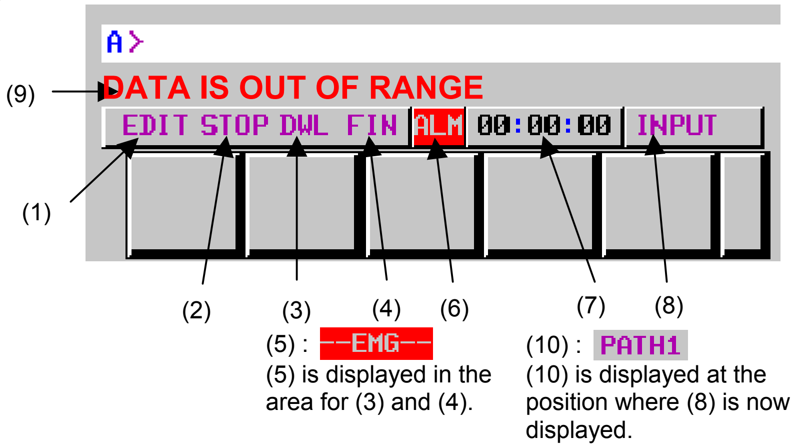


Fig. 12.6.2 (a)

(1) Current mode

MDI : Manual data input, MDI operation
MEM : Automatic operation (memory operation)
RMT : Automatic operation (DNC operation, or such like)
EDIT : Memory editing
HND : Manual handle feed
JOG : Jog feed
INC : Manual incremental feed
REF : Manual reference position return
**** : Mode other than the above

(2) Automatic operation status

**** : Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP : Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD : Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT : Automatic operation start-up (The state in which the system operates automatically)
MSTR : Manual numerical command start state (The state in which a manual numerical command is being executed)
Alternatively, tool retract and recover operation state (The state in which a recover operation and repositioning operation are being performed)

(3) Axis moving status/dwell status

MTN : Indicates that the axis is moving.
DWL : Indicates the dwell state.
*** : Indicates a state other than the above.

(4) State in which an auxiliary function is being executed

FIN : Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)
*** : Indicates a state other than the above.

(5) Emergency stop or reset status

--EMG-- : Indicates emergency stop.(Blinks in reversed display.)
--RESET-- : Indicates that the reset signal is being received.

(6) Alarm status

ALM : Indicates that an alarm is issued. (Blinks in reversed display.)
BAT : Indicates that the battery is low. (Blinks in reversed display.)
Space : Indicates a state other than the above.

> **NOTE**
> When one of the following occurs, the battery alarm state is displayed, indicating that it is time to replace the battery:
> 1 The voltage level of the lithium battery (the battery for CNC backup) becomes low.
> 2 The voltage level of the backup battery of the absolute pulse coder becomes low.

(7) Current time

hh : mm : ss - Hours, minutes, and seconds

(8) Program editing status

INPUT : Indicates that data is being input.
OUTPUT : Indicates that data is being output.
SEARCH : Indicates that a search is being performed.
EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)
LSK : Indicates that labels are skipped when data is input.
RSTR : Indicates that the program is being restarted
COMPARE : Indicates that a data comparison is being made.
OFST : Indicates that the tool length compensation amount measurement mode is set (for the machining center system) or that the tool length compensation amount write mode is set (for the lathe system).
WOFS : Indicates that the workpiece origin offset amount measurement mode is set.
AICC1 : Indicates that operation is being performed in the AI contour control I mode.
AICC2 : Indicates that operation is being performed in the AI contour control II mode.
MEM-CHK : Indicates that a program memory check is being made.
WSFT : Indicates that the workpiece shift amount write mode is set.
LEN : Indicates that the active offset value change mode (tool length offset value of the M series) is set.
RAD : Indicates that the active offset value change mode (tool radius compensation amount of the M series) is set.

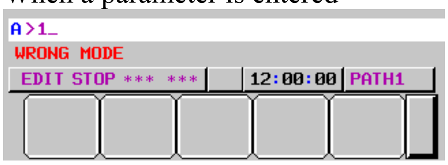
WZR : Indicates that the active offset value change mode (workpiece origin offset value) is set.
TOFS : Indicates that the active offset value change mode (tool offset value of the M series) is set.
OFSX : Indicates that the active offset value change mode (X-axis tool offset value of the T series) is set.
OFSZ : Indicates that the active offset value change mode (Z-axis tool offset value of the T series) is set.
OFSY : Indicates that the active offset value change mode (Y-axis tool offset value of the T series).
TCP : Indicates that operation is being performed in the tool center point control for 5-axis machining.
TWP : Indicates that operation is being performed in the tilted working plane command mode.
Space : Indicates that no editing operation is being performed.

(9) Warning for data setting or input/output operation

When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. In this case, the CNC does not accept the setting or input/output operation (retry the operation according to the message).

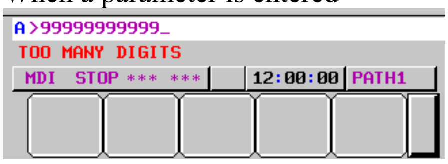
Example 1)

When a parameter is entered



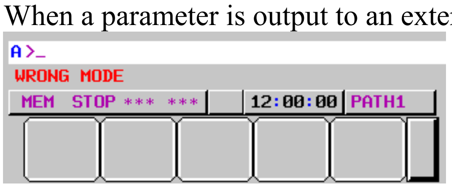
Example 2)

When a parameter is entered



Example 3)

When a parameter is output to an external input/output device



(10) Tool post name

The number of a path whose status is indicated is displayed.

PATH1 : Indicates that the status being indicated is for path 1. Other names can be used depending on the settings of parameters 3141 to 3147. The tool post name is displayed at the position where (8) is now displayed. While the program is edited, (8) is displayed.

13 GRAPHIC FUNCTION

The graphic display function can draw the tool path specified by a program being executed on a screen.
This function displays the movement of the tool during automatic operation or manual operation.

13.1 GRAPHIC DISPLAY

The tool path of a program during machining can be drawn. So, the progress of machining and the current tool position can be checked.
The following functions are available:

- The current tool position in the workpiece coordinate system is displayed.
- Graphic coordinates can be set freely.
- Rapid traverse and cutting feed can be drawn using a different color for each.
- The values of F, S, and T in the program during drawing are displayed.
- Graphic enlargement or reduction is possible.

Graphic display procedure

Explanation

Press the function key GRAPH then press the [GRAPH] soft key to display the tool path graphic screen.

- Tool path graphic screen

The tool path graphic screen consists of three major areas:

- Graphic area for drawing a tool path
- Area for displaying machining information such as tool position information
- Area for displaying a graphic coordinate system

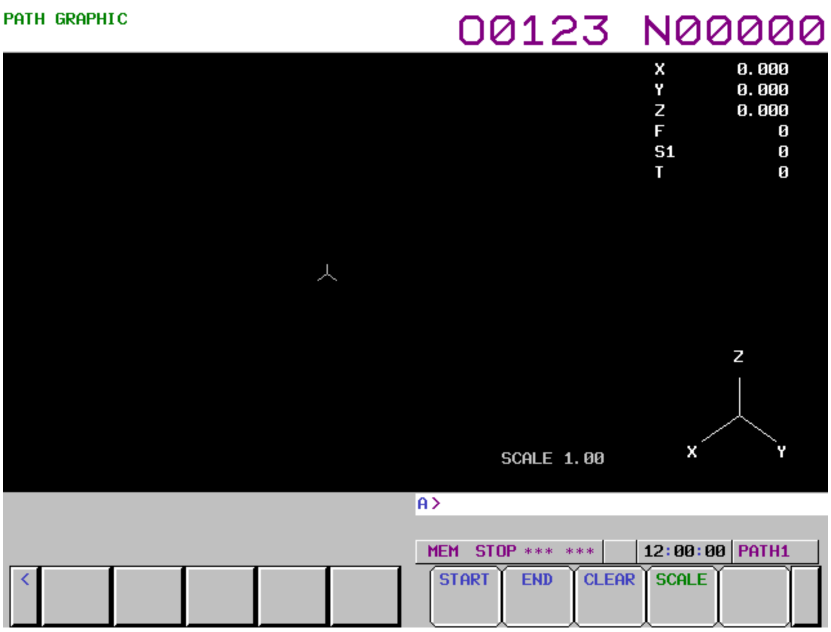


Fig. 13.1 (a) Tool path graphic screen (M series)

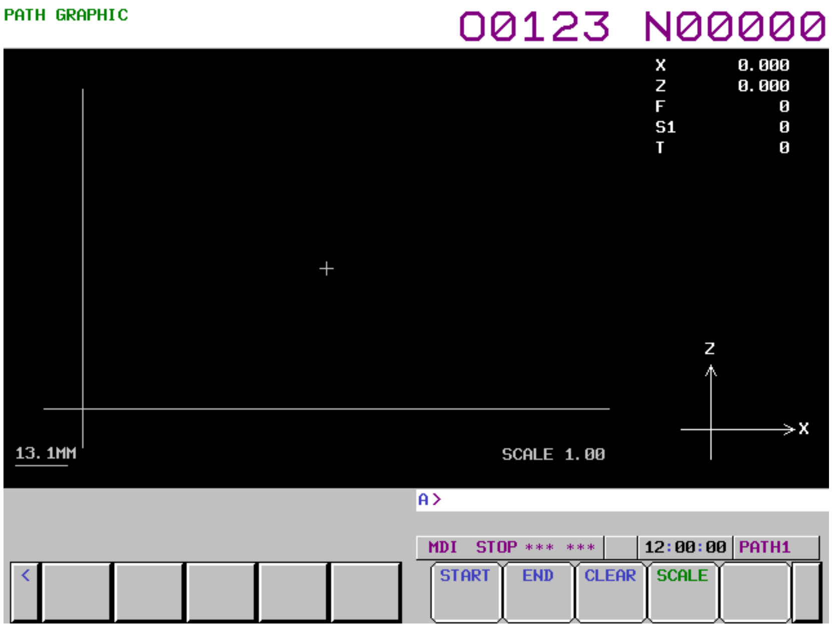


Fig. 13.1 (b) Tool path graphic screen (T series)

- Tool path

In a graphic coordinate system set by the graphic parameters described later, a tool path in the workpiece coordinate system is drawn.
Even when the tool position changes discontinuously for a cause such as the setting of the origin and the switching of the workpiece coordinate system, drawing is performed assuming that the tool has moved.
A tool path is continuously drawn even when the screen display is changed to another screen.

- Machining information

On the right side of the screen, the positions (along only three axes used for drawing) in the workpiece coordinate system, feedrate (F), spindle speed (S), and tool number (T) are displayed.

> **NOTE**
> Up to three graphic axes are used with the M series, and up to two graphic axes are used with the T series.

- Graphic coordinate system

On the lower-right portion of the screen, the coordinate axes and axis names of the graphic coordinate system are displayed.

Graphic parameter screen

Explanation

Press the function key GRAPH then press the [PARAM] soft key to display the tool path graphic screen.
On the graphic parameter screen, make settings necessary for drawing a tool path.
The graphic parameter screen consists of three pages.

M

- Graphic parameter screen page 1

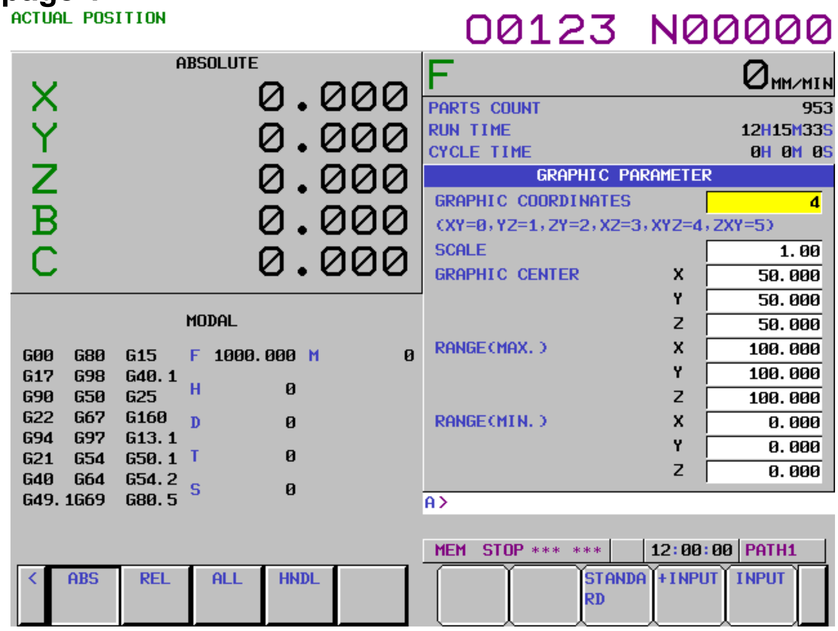


Fig. 13.1 (c) Graphic parameter screen page 1

On graphic parameter screen page 1, a graphic coordinate system, graphic range, and so forth are set.
In the setting of a graphic coordinate system, the coordinate axes and axis names of the set coordinate system are graphically displayed. When a three-dimensional coordinate system is displayed, a rotation angle is also displayed.
A graphic range is set using one of two methods. One method sets a graphic scale and graphic center coordinates. The other sets the maximum values and minimum values of a graphic range.

- Graphic parameter screen page 2

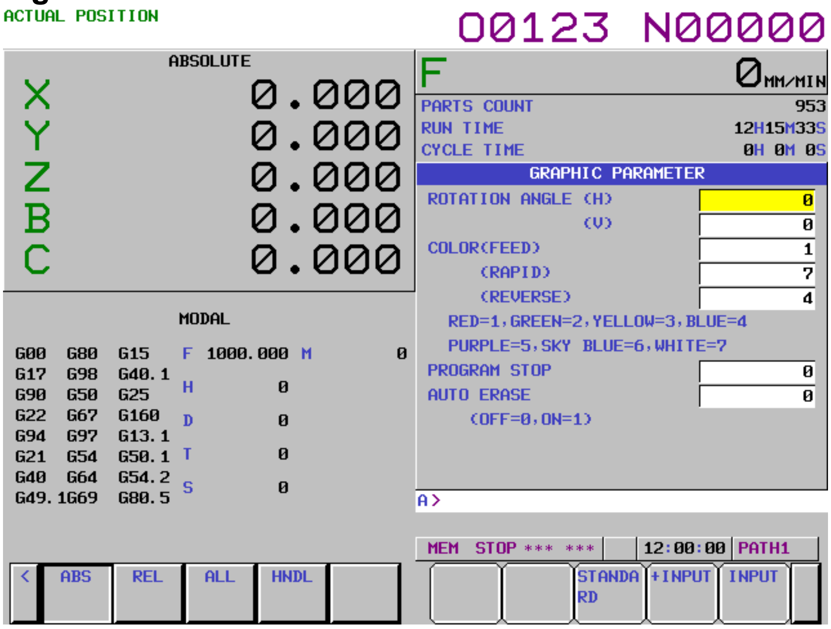


Fig. 13.1 (d) Graphic parameter screen page 2

On graphic parameter screen page 2, graphic colors, rotation angles, and whether to perform automatic erase operation are set.

- Graphic parameter screen page 3

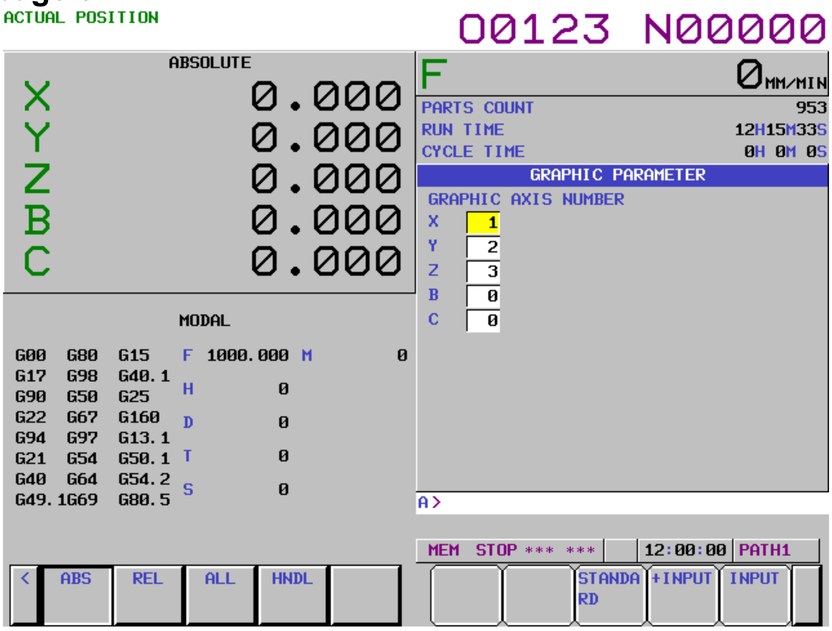


Fig. 13.1 (e) Graphic parameter screen page 3

On graphic parameter screen page 3, coordinate axes to be used for drawing are set.

T

- Graphic parameter screen page 1

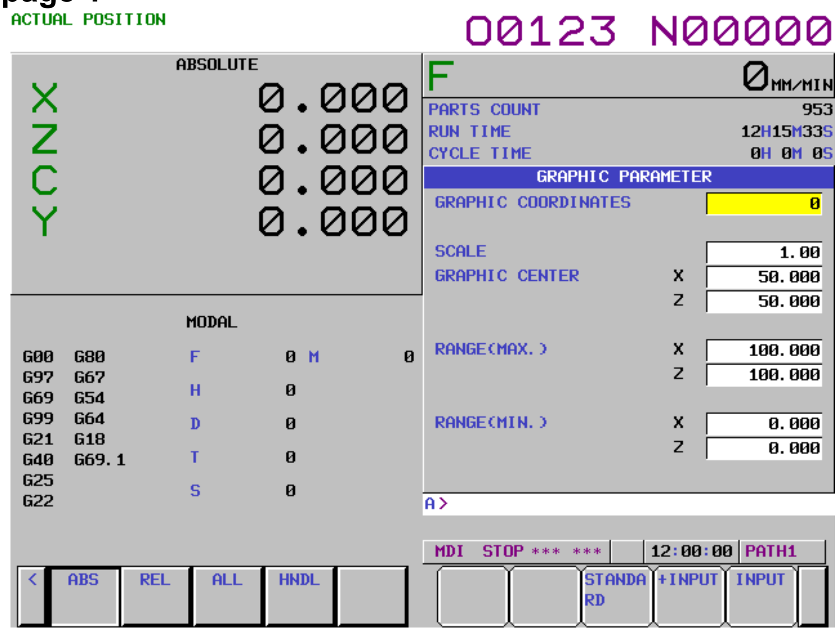


Fig. 13.1 (f) Graphic parameter screen page 1

On graphic parameter screen page 1, a graphic coordinate system, graphic range, and so forth are set.

In the setting of a graphic coordinate system, the coordinate axes and axis names of the set coordinate system are graphically displayed. When a three-dimensional coordinate system is displayed, a rotation angle is also displayed.

A graphic range is set using one of two methods. One method sets a graphic scale and graphic center coordinates. The other sets the maximum values and minimum values of a graphic range.

- Graphic parameter screen page 2

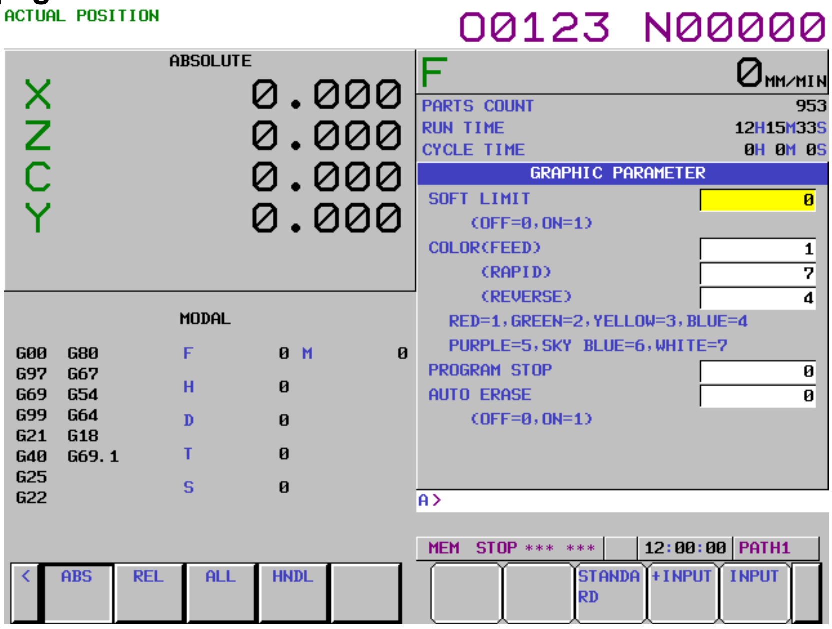


Fig. 13.1 (g) Graphic parameter screen page 2 (T series)

On graphic parameter screen page 2, graphic colors and whether to perform automatic erase operation are set.

- Graphic parameter screen page 3

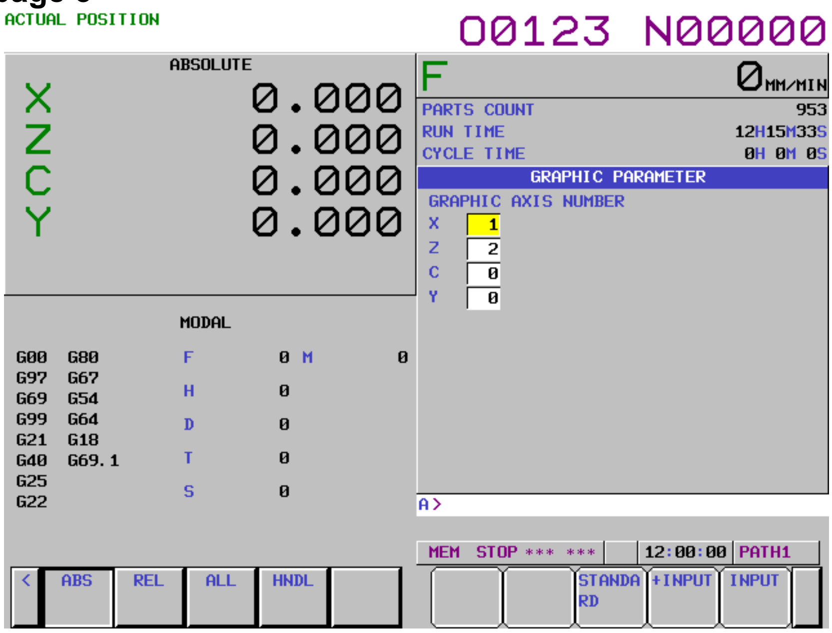


Fig. 13.1 (h) Graphic parameter screen page 3

On graphic parameter screen page 3, coordinate axes to be used for drawing are set.

Graphic parameter setting

Explanation

For tool path drawing, a graphic coordinate system, tool path graphic colors, and graphic range need to be set on the graphic parameter screen.
The graphic parameters to be set on the graphic parameter screen are described below.
When a value is set for a parameter, the parameter value becomes immediately effective. If a tool path is already drawn, the tool path is cleared when new parameter values are set.

- Graphic coordinate system

Select a desired graphic coordinate system for tool path drawing then set the corresponding number.

M

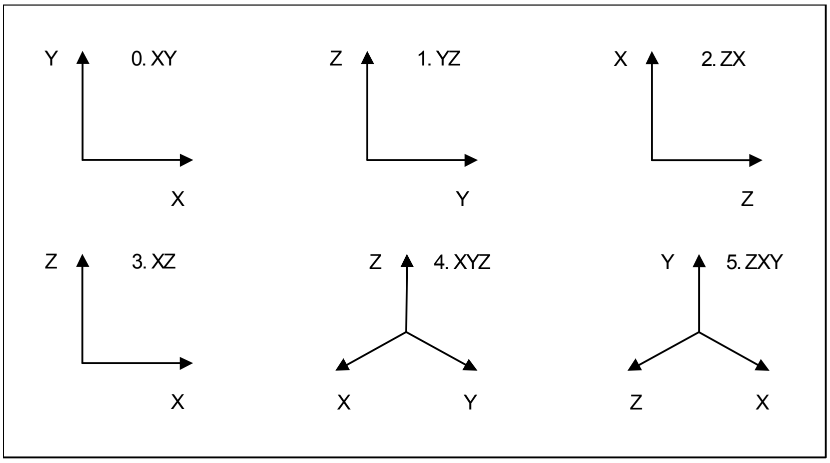


Fig. 13.1 (i) Graphic coordinate system (M series)

T

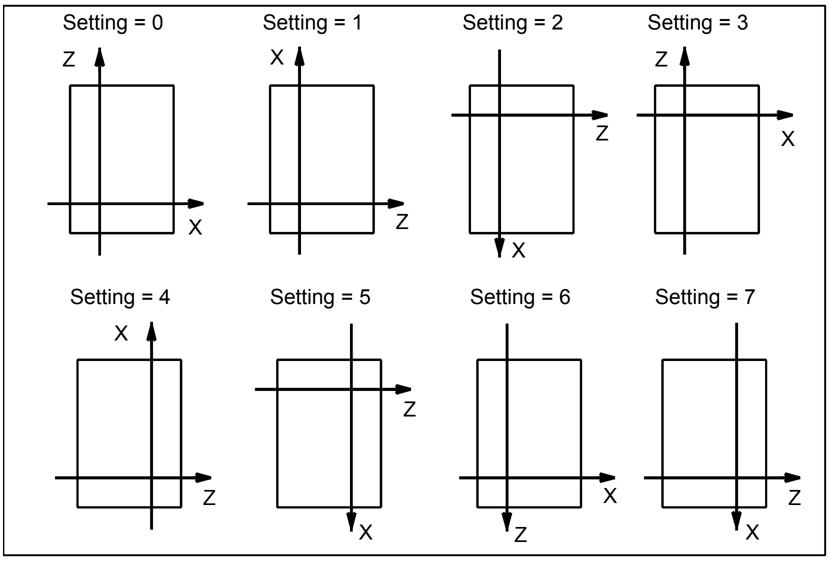


Fig. 13.1 (j) Graphic coordinate system (T series)

M

- Horizontal rotation angle

When a three-dimensional graphic coordinate system such as 4.XYZ or 5.ZXY is selected, the coordinate system can be rotated with the horizontal plane used as the rotation plane. Set a rotation angle from -360° to +360°.
In Fig. 13.1 (k) below, the graphic coordinate system XYZ is converted to XY"Z" by the following settings:

Initial rotation angle: 180°
Horizontal rotation angle: 30°

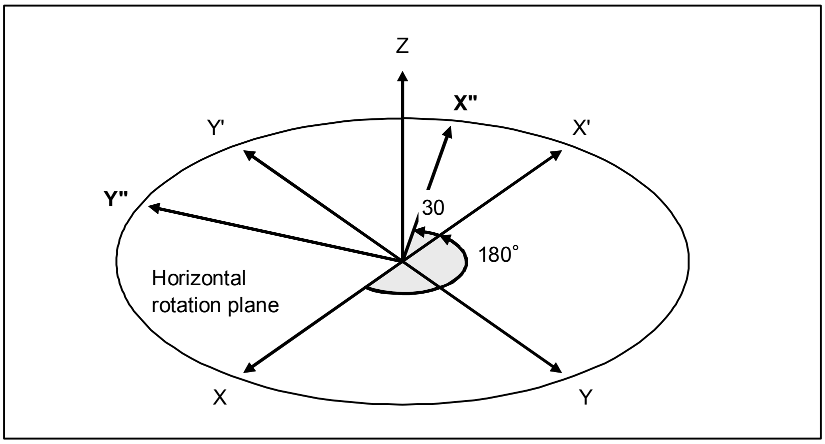


Fig. 13.1 (k) Coordinate system rotation in horizontal direction

- Vertical rotation angle

When a three-dimensional coordinate system such as 4.XYZ or 5.ZXY is selected, the coordinate system can be rotated with an axis on the horizontal plane specified as a vertical rotation axis. Set a rotation angle from -360° to +360°.
In Fig. 13.1 (l) below, the graphic coordinate system XYZ is converted to X'Y'Z' by the following settings:

Rotation angle of the vertical rotation axis: 65°
Vertical rotation angle: 20°

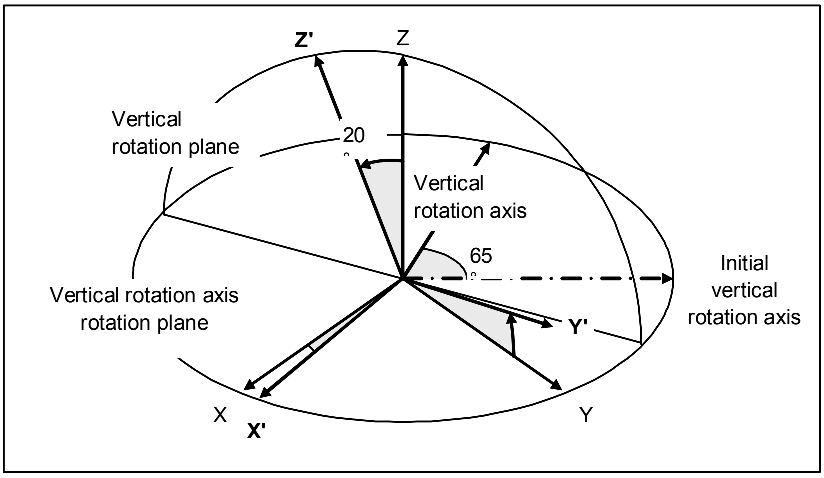


Fig. 13.1 (l) Coordinate system rotation in vertical direction

- Graphic color

Set a graphic color number for a tool path for each of cutting feed and rapid traverse.

1: Red 2: Green 3: Yellow 4: Blue
5: Purple 6: Sky blue 7: White

- Graphic range setting

Set a graphic range so that a tool path can be drawn in the tool path graphic area. Two methods are available:

1. Method that sets graphic center coordinates and a scale
2. Method that sets the maximum values and minimum values of a graphic range

Whether method 1 or method 2 is used is determined by which parameters were set most recently. A set graphic range is preserved even if the power is turned off.

Method that sets graphic center coordinates and a scale

Set the coordinates of the center of a tool path graphic area in the workpiece coordinate system. Then, set a scale used to contain the graphic range in the tool path graphic area.
Set a scale value from 0.01 to 100 (magnification).
As a smaller scale is set, a tool path can be drawn in a wider range. As a larger scale is set, a tool path is drawn by enlarging the tool path around the graphic center coordinates.

Method that sets the maximum values and minimum values of a graphic range

Set the maximum coordinates and minimum coordinates of a desired graphic range in the workpiece coordinate system. Drawing is performed so that the specified entire range is contained in the tool path graphic area.

From set maximum values and minimum values, graphic center coordinates and a sale are automatically calculated to update the graphic center coordinates and scale on the graphic parameter screen.
When a scale is automatically determined, the scale is clamped to within the range 0.01 to 100. Moreover, a maximum value must be greater than the corresponding minimum value.

> **NOTE**
> When the maximum values and minimum values of a graphic range are set, the graphic center coordinates and scale are automatically updated. However, when the graphic center coordinates and scale are changed, the maximum values and minimum values of the graphic range are not automatically updated.

- Automatic erasure

Before drawing is started, the previous drawing can be erased automatically.

1: Immediately before drawing is started, the previous drawing is erased automatically.

0: The previous drawing is not erased automatically.

- Graphic axis number

Set which controlled axis to be assigned to which graphic axis.
For each controlled axis, set one of the following graphic axis numbers:

First graphic axis: 1
Second graphic axis: 2
Third graphic axis: 3
Axis not used for drawing: 0

NOTE

1 When 0 is set for all controlled axes, it is assumed that 1, 2, and 3 are set sequentially for the first to the third controlled axes.

2 With the T series, a tool path is drawn along the first and second graphic axes. No tool path is drawn along the third graphic axis.

Operation for graphic parameter setting

Operation

- Moving the cursor

The cursor can be moved to a desired parameter by the page key [↑ PAGE] or [PAGE ↓] and the cursor key [↑], [↓], [←], or [→].
With the cursor keys, however, you cannot move from page 1 or 2 to page 3.

- Input of settings (absolute input)

Method 1

(1) Key in a value to be set.
(2) Press the [INPUT] soft key.

Method 2

(1) Key in a value to be set.
(2) Press the [INPUT] key.

- Input of settings (incremental input)

Method 1

(1) Key in a value to be incremented to or decremented from the current setting.
(2) Press the [+INPUT] soft key.

Procedure for tool path drawing

Procedure

- Start of drawing

(1) Display the tool path graphic screen.
(2) Press the [START] soft key.
The state that enables the movement of the tool in automatic operation or manual operation to be drawn is set.
Afterwards, tool path drawing continues even if another screen is displayed.
(3) Start automatic operation or manual operation.

- End of drawing

(1) Display the tool path graphic screen.
(2) Press the [END] soft key.
Tool path drawing ends.

- Erasure of the drawing

(1) Press the [END] soft key to end drawing.
(2) Press the [CLEAR] soft key. The tool path drawn so far is erased.

> **NOTE**
> 1 Set the machine lock state to perform drawing only without moving the tool.
> 2 When the feedrate is high, the tool path may not be drawn correctly. In such a case, decrease the feedrate by performing, for example, a dry run.

Enlarged/reduced display

On the tool path graphic screen, you can move the center position of the tool path drawing or enlarge the tool path drawing while viewing the drawn tool path.
If any of these operations is executed, the tool path already drawn is cleared.

- Procedure for changing the graphic range by setting a graphic center and magnification

The center position of drawing can be moved. At the same time, the scale can also be changed. So, the tool path can be enlarged or reduced at a desired new center position.

(1) Press the [SCALE] soft key then the [CENTER MAGNIFICATION] soft key.
A yellow cursor appears at the center of the screen, and the soft key display is changed.
(2) Move the yellow cursor to a new graphic center position by using the cursor key ↑, ↓, ←, or →.
(3) When changing the scale, key in a value from 0.01 to 100 (magnification) then press the [INPUT] soft key. An input value is displayed at "SCALE" in the lower-right corner of the screen.
When you press the [+INPUT] soft key, the current magnification is incremented by an input value.
(4) Press the [EXEC] soft key to end the operation.
After this step, the setting for graphic movement is effective to enable drawing with the new setting.

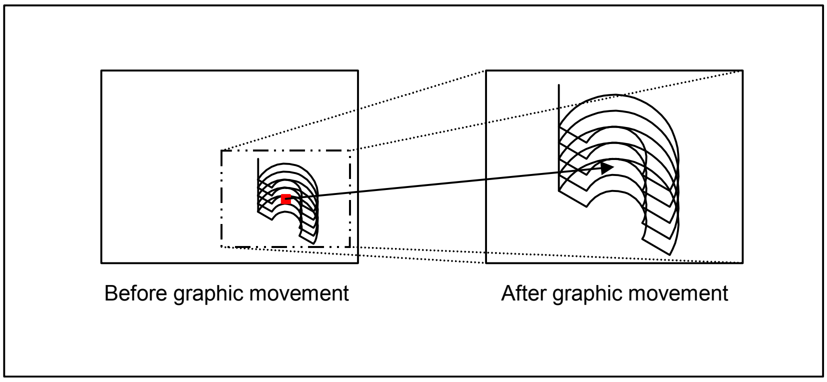


Fig. 13.1 (m) Graphic movement (magnification = 2.00)

- Procedure for changing the graphic range with a rectangle

A tool path can be drawn by enlarging a specified rectangular area.

(1) Press the [SCALE] soft key then the [RECTANGLE] soft key.
Two cursors, one in red and the other in yellow, appear at the center of the screen, and the soft key display is changed.

(2) Move the yellow cursor by using the cursor key ↑, ↓, ←, or →. The cursor to be moved can be switched by pressing the [CURSOR UP/DOWN] soft key.
Move the two cursors to the diagonal points of a new rectangular graphic range. A tool path is drawn next time so that the drawn tool path is contained in this rectangular range.

(3) Press the [EXEC] soft key to end the operation.
After this step, the setting of graphic enlargement is effective to enable drawing with the new setting.

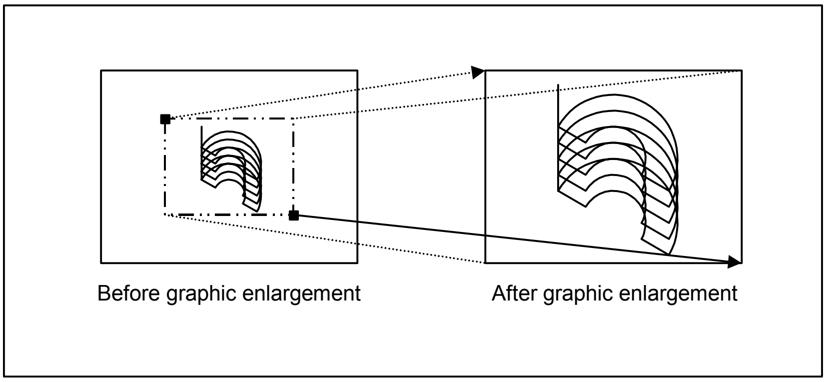


Fig. 13.1 (n) Graphic enlargement

> **NOTE**
> 1 To stop an enlargement/reduction operation, press the [CANCEL] soft key.
> 2 Even if you perform an enlargement/reduction operation, the tool path already drawn on the screen is neither moved nor enlarged. The setting for enlargement/reduction becomes effective starting with tool path drawing after you press the [EXEC] soft key.

IV. MAINTENANCE

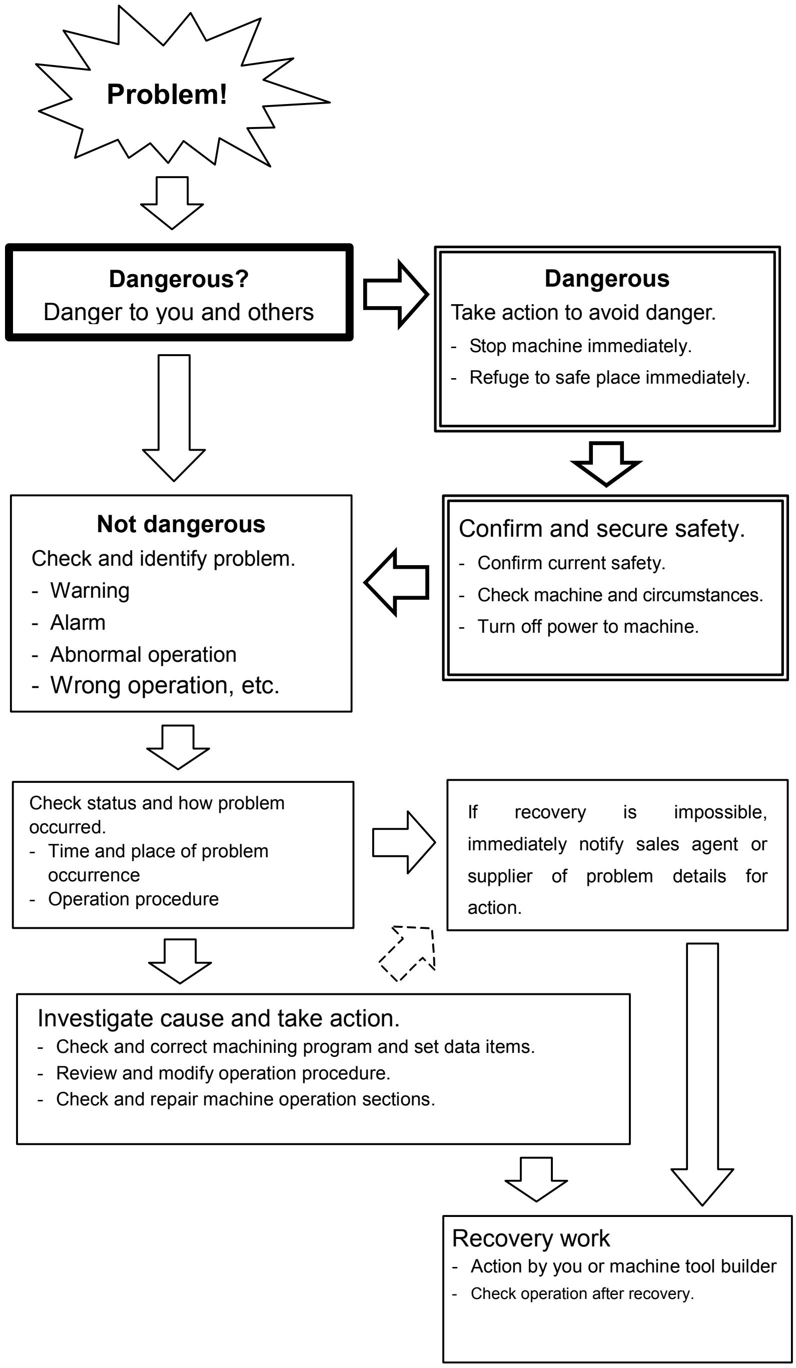
1 ROUTINE MAINTENANCE

This chapter describes routine maintenance work that the operator can perform when using the CNC.

> ⚠ **WARNING**
> Only those persons who have been educated for maintenance and safety may perform maintenance work not described in this chapter.

1.1 ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED

If an unexpected operation occurs or an alarm or warning is output when the CNC and machine are used, the problem needs to be solved quickly. For this purpose, the status of the problem must be identified correctly, and a proper action must be taken.
The procedure for taking an action for a problem is shown below.



For details of investigation and action on problems arising from the CNC, refer to "TROUBLESHOOTING PROCEDURE" in the maintenance manual issued by FANUC.

1.2 BACKING UP VARIOUS DATA ITEMS

With the CNC, various data items such as machining programs, offset data, and system parameters are stored in the SRAM of the control unit and are protected by a backup battery. However, an accident can erase the data. By storing the data at another location (outside the CNC), the data, when lost, can be restored.
So, when the machine is started up or data is updated, for example, the data should be backed up (stored outside the CNC).

- Data backup operation

The data items listed below should be backed up. For the method of data output operation, see the chapter of "DATA INPUT/OUTPUT" in this manual.

<1> System parameters
→ See III-8.1.2.
<2> Machining programs
→ See III-8.1.1.
<3> Tool offset data
→ See III-8.1.3.
<4> PMC data
→ See PMC PROGRAMMING MANUAL (B-63983EN).
<5> Pitch error compensation data (when the pitch error compensation function is selected.)
→ See III-8.1.4.
<6> Three-dimensional error compensation (when the three-dimensional error compensation function is selected.)
→ See III-8.1.5.
<7> Custom macro variables (when the custom macro function is selected.)
→ See III-8.1.6.
<8> Workpiece coordinate system setting data (when the workpiece coordinate system function is selected.)
→ See III-8.1.7.
<9> Tool management data (when the tool management function is selected.)
→ See III-8.1.9.

It is recommended that recording media (such as floppy disks and memory cards) daily used with the machine be used to store data. Stored data should be managed properly so that the data can be restored quickly if a problem occurs.

- Data restoration work

In order to restore lost data to the state of the stored data, input the data backed up according to the previous item into the CNC. For the method of data input operation, see the chapter of "DATA INPUT/OUTPUT" in this manual.

> ⚠ **WARNING**
> After inputting stored data, do not start an operation immediately. Instead, check that the data is input correctly and that settings are made to meet a desired operation.
> If an operation is executed without making this check, the machine and workpiece can be damaged and personal injury can occur due to an unexpected machine movement. Use sufficient care.

> ⚠ **CAUTION**
> Before recovery of the following data items, consult with the machine tool builder of the machine used:
> - System parameters
> - PMC data
> - Macro programs and custom macro variables
> - Pitch error compensation values

> **NOTE**
> The method of recovery described in this section is intended just to restore the state of the backed up data, and does not guarantee recovery of the state that was present when the data was lost.

1.3 METHOD OF REPLACING BATTERY

This chapter describes how to replace the CNC backup battery and absolute Pulsecoder battery. This chapter consists of the following sections:

Battery for memory backup

Part programs, offset data, and system parameters are stored in SRAM in the control unit. The power to the SRAM is backed up by a lithium battery mounted on the front panel of the control unit. Therefore, the above data is not lost even if the main battery fails. The backup battery is installed in the control unit prior to being shipped from the factory. This battery can provide backup for the memory contents for about a year.

When the battery voltage falls, alarm message "BAT" blinks on the LCD display and the battery alarm signal is output to the PMC. When this alarm is displayed, replace the battery as soon as possible. In general, the battery can be replaced within one or two weeks of the alarm first being issued. This, however, depends on the system configuration.

If the battery voltage subsequently drops further, backup of memory can no longer be provided. Turning on the power to the control unit in this state causes system alarm to be issued because the contents of memory are lost. Replace the battery, clear the entire memory, then reenter the data.

Replace the memory backup battery while the control unit is brought off.

The following two kinds of batteries can be used.

- Lithium battery, incorporated into the CNC control unit.
- Two alkaline dry cells (size D) in an external battery case.

NOTE
A lithium battery is installed as standard at the factory.

1.3.1 Replacing Battery for LCD-mounted Type CNC Control Unit

When using a lithium battery

- Replacement procedure

When a lithium battery is used

Prepare a new lithium battery (ordering code: A02B-0200-K102 (FANUC specification: A98L-0031-0012)).

<1> Turn on the power to the CNC. After about 30 seconds, turn off the power.

<2> Remove the old battery from the back of the CNC control unit.
First, unplug the connector by yanking the battery cable, then take the battery out of its case.
The battery case of a control unit without option slots is located at the back of the unit as shown in the figure below. The battery case of a control unit with option slots is located next to the fan on the top of the unit.

<3> Insert a new battery and reconnect the connector.

<4> Clamp the battery cable as shown in Fig. 1.3.1 (c).

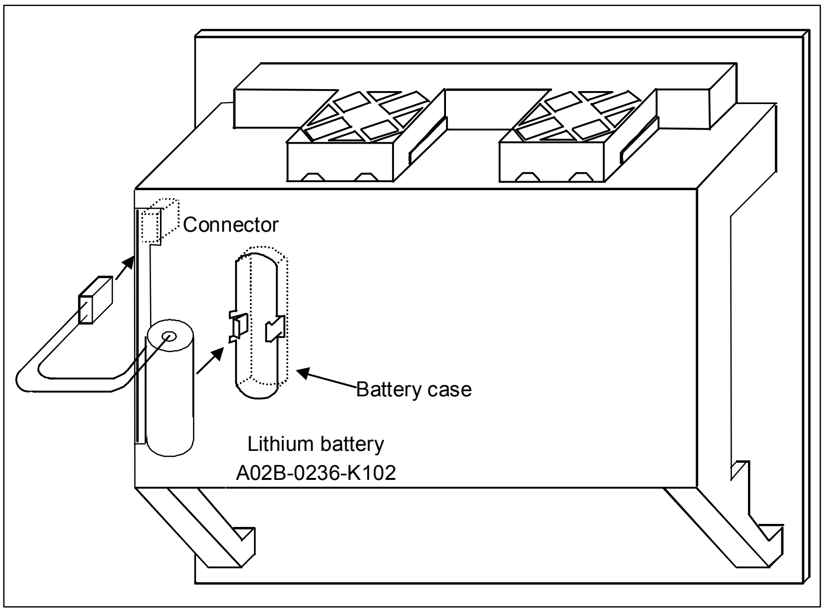


Fig. 1.3.1 (a) Unit without option slots

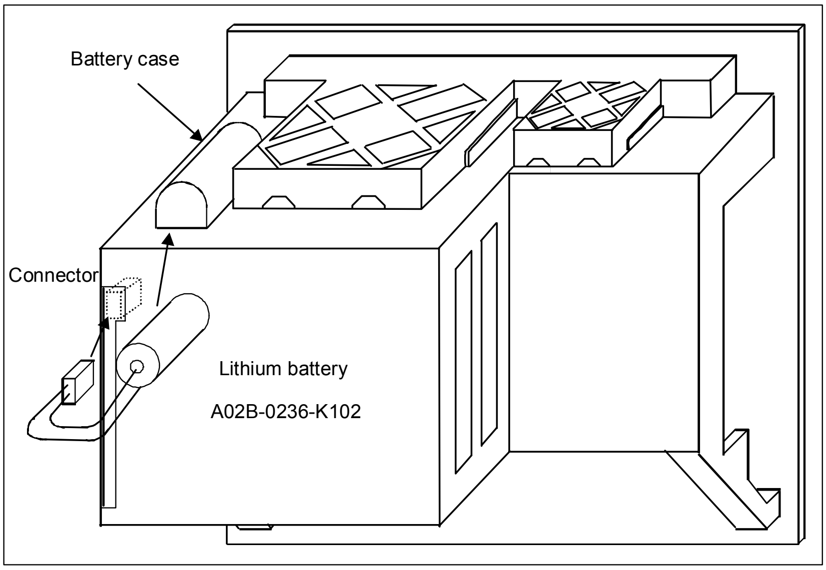


Fig. 1.3.1 (b) Unit with option slots

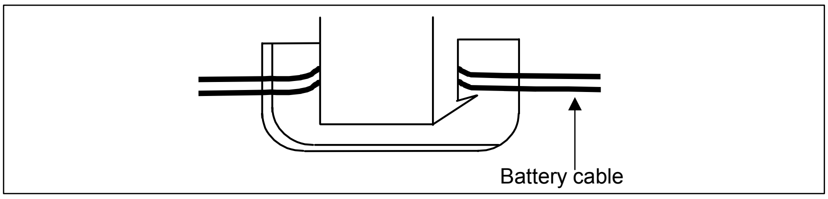


Fig. 1.3.1 (c) Clamping the battery cable

> ⚠ **WARNING**
> Using other than the recommended battery may result in the battery exploding. Replace the battery only with the specified battery (A02B-0200-K102).

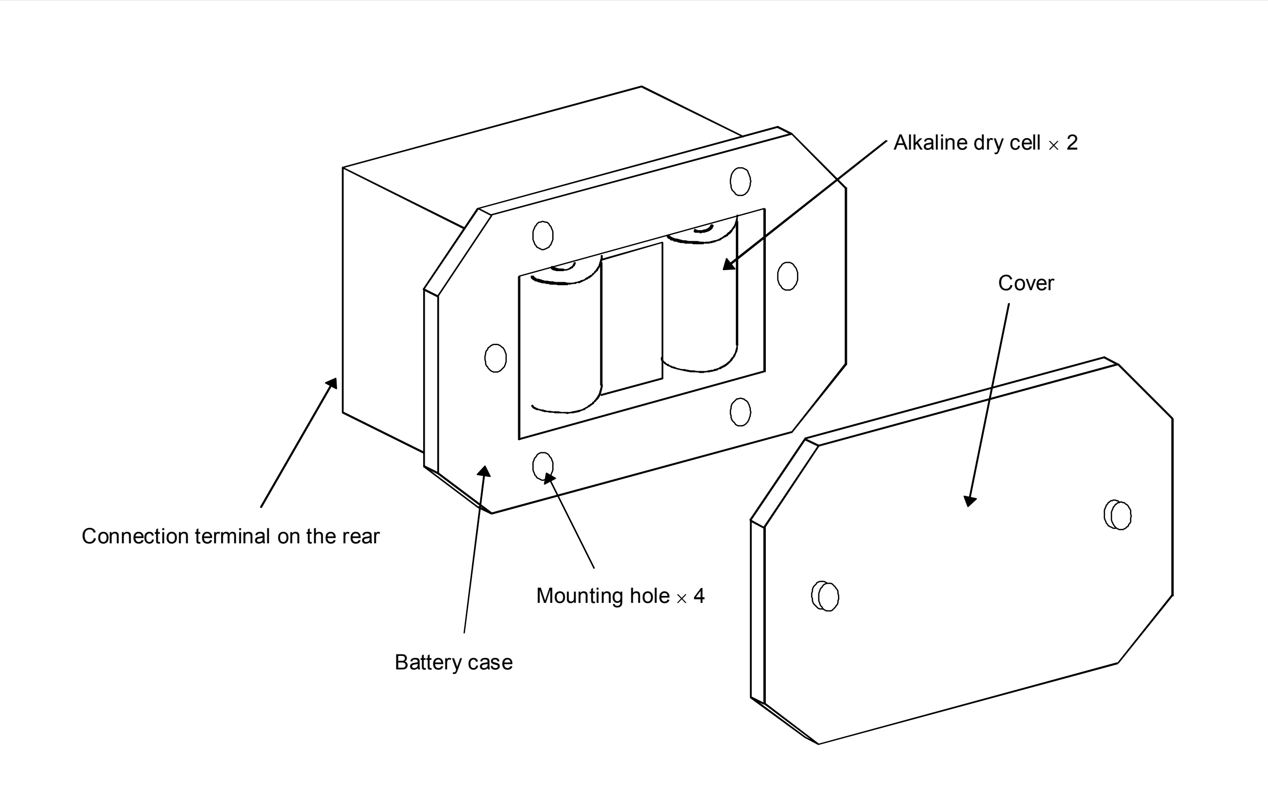
> ⚠ **CAUTION**
> Steps <1> to <3> should be completed within 30 minutes.
> Do not leave the control unit without a battery for any longer than the specified period. Otherwise, the contents of memory may be lost.
> If steps <1> to <3> may not be completed within 30 minutes, save all contents of the SRAM to the memory card beforehand. Thus, if the contents of the SRAM are lost, the contents can be restored easily.
> For the method of operation, refer to Maintenance Manual.

When discarding a battery, observe the applicable ordinances or other rules of your local government. Also, cover the terminals of the battery with vinyl tape or the like to prevent a short-circuit.

When using commercial alkaline dry cells (size D)
- Replacement procedure

<1> Prepare two alkaline dry cells (size D) commercially available.
<2> Turn on the power to the control unit.
<3> Remove the battery case cover.
<4> Replace the cells, paying careful attention to their orientation.
<5> Reinstall the cover onto the battery case.

> ⚠ **CAUTION**
> When replacing the alkaline dry cells while the power is off, use the same procedure as that for lithium battery replacement described above.



1.3.2 Replacing the Battery for Stand-alone Type CNC Control Unit

When using a lithium battery

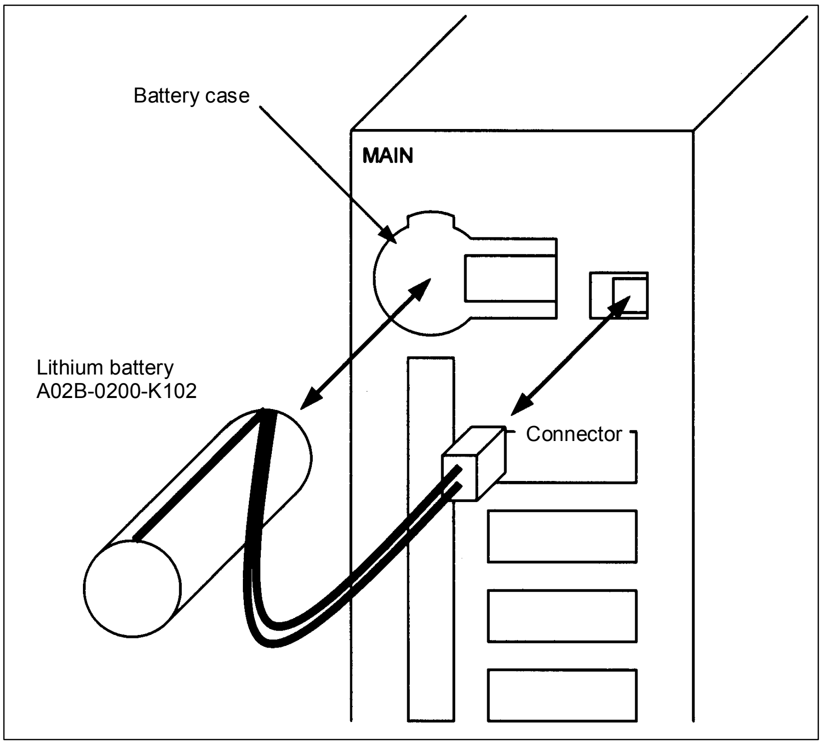
- Replacing the battery

If a lithium battery is used, have A02B-0200-K102 (FANUC internal code: A98L-0031-0012) handy.

<1> Turn the CNC on. About 30 seconds later, turn the CNC off.

<2> Remove the battery from the top area of the CNC unit.
First, unplug the connector by yanking the battery cable, then take the battery out of the case.
The battery case is provided in the top area of the face plate of the main CPU board.

<3> Replace the battery, then connect the connector.



> ⚠ **WARNING**
> The incorrect mounting of the battery may cause an explosion. Avoid using any battery other than the one specified here (A02B-0200-K102).

> ⚠ **CAUTION**
> Complete steps <1> to <3> within 30 minutes.
> If the battery is left removed for a long time, the memory would lose the contents.
> If there is a danger that the replacement cannot be completed within 30 minutes, save the whole contents of the SRAM to a memory card. The contents of the memory can be easily restored with the memory card in case the memory loses the contents.

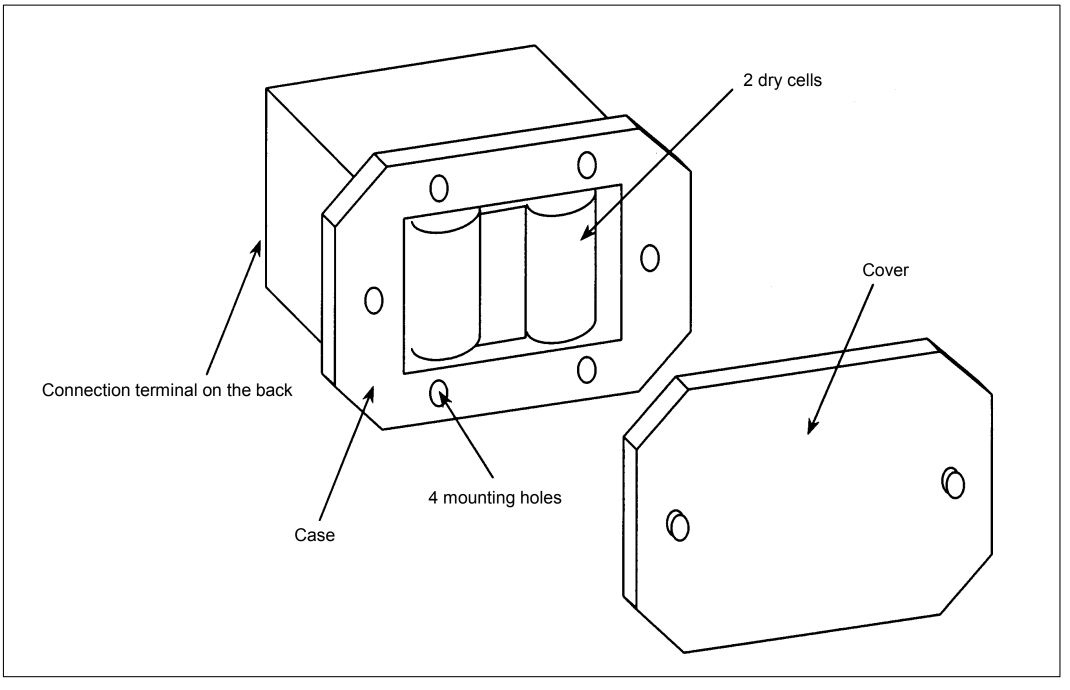
Discard the dead battery, observing appropriate municipal rules and regulations. When discarding the battery, insulate the terminal with a tape so that no short-circuit would occur.

When using commercial D-size alkaline dry cells

- Replacing the battery

<1> Have commercial D-size alkaline dry cells handy.
<2> Turn the CNC on.
<3> Remove the cover from the battery case.
<4> Replace the old dry cells with new ones. Mount the dry cells in a correct orientation.
<5> Replace the cover on the battery case.

> ⚠ **CAUTION**
> In the power-off state, the battery should be replaced as in the case of the lithium battery, which is descried above.



1.3.3 Battery in the CNC Display Unit with PC Functions (3 VDC)

A lithium battery is used to back up BIOS data in the CNC display unit with PC functions. This battery is factory-set in the CNC display unit with PC functions. This battery has sufficient capacity to retain BIOS data for one year.
When the battery voltage becomes low, the following message appears on the self test screen displayed at turn-on, and the self test pauses.

CMOS Battery Failure

If this message appears, replace the battery as soon as possible (within one week). FANUC recommends that the battery be replaced once per year regardless of whether a battery alarm is issued.

- Replacing the battery

(1) To guard against the possible loss or destruction of BIOS parameters, write down the BIOS parameter values.
(2) Obtain a new lithium battery (A02B-0200-K102).
(3) After power has been supplied for at least five seconds, turn off the power to CNC display unit with PC functions. Remove the intelligent terminal from the panel so that replacement work can be done from the rear of the intelligent terminal.
(4) Detach the connector of the lithium battery, and remove the battery from the battery holder.
(5) Attach the connector, and place the battery in the battery holder.
(6) Install CNC display unit with PC functions again.
(7) Turn on the power, and check that the BIOS parameters are maintained (BIOS setup is not activated forcibly).

Between removing an old battery and inserting new battery, no more than five minutes must be allowed to elapse.

When replacing the battery of the PANEL *i*, follow the steps above.

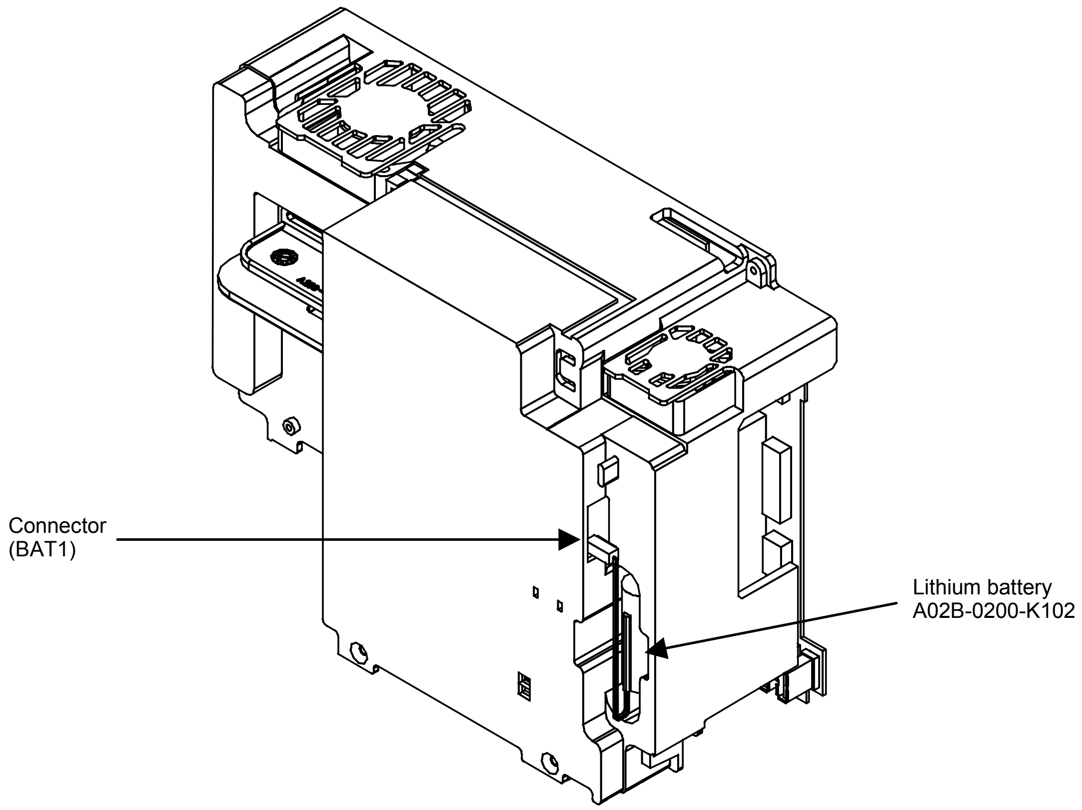


Fig. 1.3.3 (a) Lithium battery connection for CNC display unit with PC functions

1.3.4 Battery for Absolute Pulsecoders

(1) When the voltage of the battery for absolute Pulsecoders becomes low, alarms DS0306 to DS0308 occur.
(2) When alarm DS0307 (alarm indicating the voltage of the battery becomes low) occurs, replace the battery as soon as possible. In general, the battery should be replaced within one or two weeks, however, this depends on the number of Pulsecoders used.
(3) If the voltage of the battery becomes any lower, alarm DS0306 (battery zero alarm) occurs. In this case, the current positions for the Pulsecoders can no longer be maintained.
In this state, alarm DS0300 (reference position return request alarm) occurs. After replacing the batteries, be sure to perform manual reference position return.
(4) The service life of the batteries is about two years if they are used in a six-axis configuration with α*i*/α*i*s/β*i*s series servo motors and one year if they are used in a six-axis configuration with α/β series servo motors. FANUC recommends that you replace the batteries periodically according to the battery service life.
(5) To connect the battery, use the battery case or incorporate the battery into the servo amplifier. Note that the attachment method of the battery depends on the connection method and the type of servo amplifier.

- Replacing batteries

To prevent absolute position information in absolute Pulsecoders from being lost, turn on the machine power before replacing the battery. The replacement procedure is described below. (Note: The turning-on step is not required when the α*i* or α*i*s series servo motor or β*i*s series servo motor (β0.4*i*s to β22*i*s) is used.)

(1) Turn the servo unit (machine) on.
(2) Place the machine in the emergency stop state.
(3) Confirm that servo motors are not active.
(4) Make sure the DC link charge LED is off.
(5) Remove the old battery and then attach a new one.
(6) Now, replacement has been completed. The system power can be turned off.

NOTE

The absolute Pulsecoder of the servo motor α*i*/α*i*s series or β*i*s (β0.4*i*s to β22*i*s) series is incorporated with a backup capacitor as standard. This backup capacitor enables an absolute position detection to be continued for about 10 minutes. Therefore, no manual reference position return need be performed if the time during which servo amplifier power is kept off for battery replacement is within 10 minutes. If battery replacement takes 10 minutes or more, the power must remain turned on.

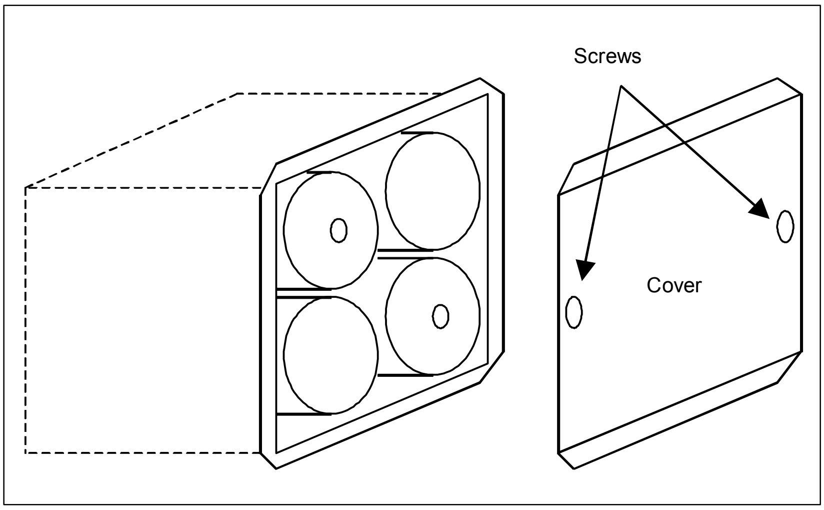
⚠ **WARNING**

1. When replacing the battery, be careful not to touch bare metal parts in the panel. In particular, be careful not to touch any high-voltage circuits due to the electric shock hazard.
2. Before replacing the battery, make sure the DC link charge LED is off. Otherwise, an electric shock may be received.
3. Be sure to use the specified battery. If another type of battery is used, it may overheat, blow out, or catch fire.
4. Install the battery with correct polarity. If the battery is installed with incorrect polarity, it may overheat, blow out, or catch fire. Or, absolute position information in absolute Pulsecoders may be lost.
5. During attachment of the battery, insert the factory-attached protection socket into the CX5X or CX5Y connector, whichever is not used. If the +6 V pin and 0 V pin are short-circuited, the battery may overheat, blow out, or catch fire. Or, absolute position information in absolute Pulsecoders may be lost.

- Replacing D-size alkaline dry cells in the battery case

Replace four D-size alkaline batteries (A06B-6050-K061) in the battery case installed in the machine.

(1) Have four D-size alkaline batteries on hand.
(2) Loosen the screws on the battery case. Remove the cover.
(3) Replace the alkaline batteries in the case. Pay careful attention to the polarity of the alkaline batteries.
(4) Attach the cover.



⚠ **WARNING**

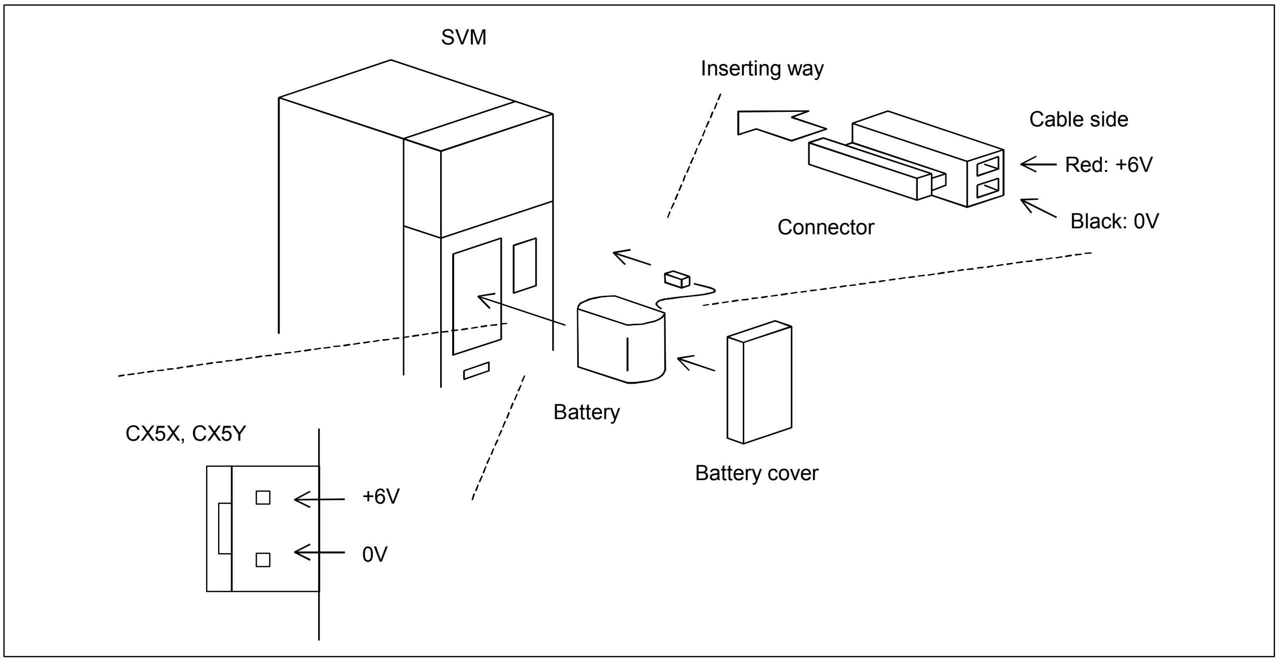
Install the battery with correct polarity. If the battery is installed with incorrect polarity, it may overheat, blow out, or catch fire. Or, absolute position information in absolute Pulsecoders may be lost.

- Attaching the built-in battery (αi series servo amplifier)

Attach the lithium battery (A06B-6073-K001) to the servo amplifier.

[Attachment procedure]

(1) Remove a battery cover from the servo amplifier.
(2) Attach the battery as shown below.
(3) Re-attach the cover.
(4) Connect the connector of battery with CX5X of the servo amplifier.



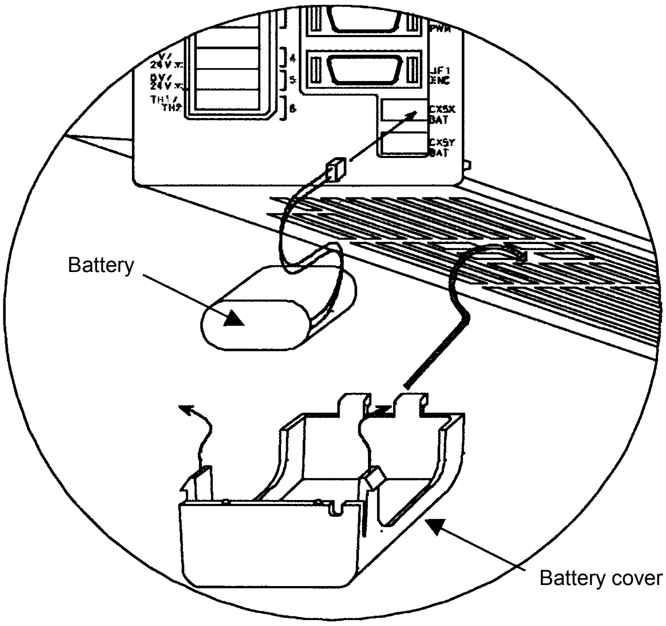
⚠ CAUTION
Attaching the battery from the cable outlet applies tension to the cable. Therefore, attach the cable from another place to prevent the cable from being stretched. If this cable is connected on a stretch condition, a bad conductivity may be occurred.

- Attaching the built-in battery (β series servo amplifier)

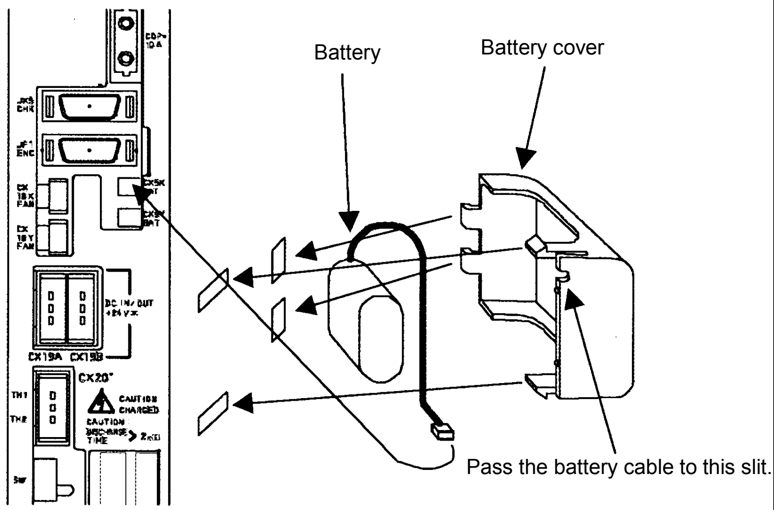
Attach the lithium battery (A06B-6093-K001) to the servo amplifier.

[Attachment procedure]

(1) In case of SVU-12 or SVU-20, remove the battery cover under the servo amplifier grasping its left and right sides. In case of SVU-40 or SVU-80, remove the cover attached on right side of the servo amplifier grasping its upper and lower sides.
(2) Remove the connector of battery. (Connector CX5X or CX5Y)
(3) Replace the battery and connect the connector.
(4) Mount the battery cover.



SVU-12, SVU-20



SVU-40, SVU-80

> ⚠ **CAUTION**
> 1 The connector of the battery can be connected with either of CX5X and CX5Y.
> 2 Attaching the battery from the cable outlet applies tension to the cable. Therefore, attach the cable from another place to prevent the cable from being stretched. If this cable is connected on a stretch condition, a bad conductivity may be occurred.

- Used batteries

Old batteries should be disposed as "INDUSTRIAL WASTES" according to the regulations of the country or autonomy where your machine has been installed.

APPENDIX

A PARAMETERS

This manual describes all parameters indicated in this manual.
For those parameters that are not indicated in this manual and other parameters, refer to the parameter manual.

NOTE

A parameter that is valid with only one of the path control types for the lathe system (T series) and machining center system (M series) is indicated in the upper or lower row as described below. A blank represents an unusable parameter.

[Example 1]

The parameter HTG is common to the T series and M series, and RTV and ROC are parameters used with the T series only.

	#7	#6	#5	#4	#3	#2	#1	#0	
1403	RTV		HTG	ROC					T series
			HTG						M series

[Example 2]

The following parameter is used with the M series only:

1411		T series
	Cutting feedrate	M series

A.1 DESCRIPTION OF PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
0000							ISO	TVC

[Input type] Setting input
[Data type] Bit path

 # 0 TVC TV check
0: Not performed
1: Performed

 # 1 ISO Code used for data output
0: EIA code
1: ISO code

> **NOTE**
> ASCII code is used at all times for output to the memory card.

	#7	#6	#5	#4	#3	#2	#1	#0
0001							FCV	

[Input type] Setting input
[Data type] Bit path

 # 1 FCV Program format
0: Series 16 standard format
1: Series 15 format

	#7	#6	#5	#4	#3	#2	#1	#0
0010							PRM	

[Input type] Setting input
[Data type] Bit path

 # 1 PRM When parameters are output, the parameters whose values are 0 are:
0: Output.
1: Not output.

	#7	#6	#5	#4	#3	#2	#1	#0
0012								MIRx

[Input type] Setting input
[Data type] Bit axis

 # 0 MIRx Mirror image for each axis
0: Mirror image is off. (Normal)
1: Mirror image is on. (Mirror)

0020	I/O CHANNEL : Input/output device selection, or interface number for a foreground input device

[Input type] Setting input
[Data type] Byte
[Valid data range] 0 to 5

The CNC has the following interfaces for transferring data to and from an external input/output device and the host computer:

Input/output device interface (RS-232-C serial ports 1 and 2)
Memory card interface
Data server interface

By setting bit 0 (IO4) of parameter No. 0110, data input/output can be controlled separately. When IO4 is not set, data input/output is performed using the channel set in parameter No. 0020. When IO4 is set, a channel can be assigned to each of foreground input, foreground output, background input, and background output.
In these parameters, specify the interface connected to each input/output device to and from which data is to be transferred. See the table below for these settings.

Correspondence between settings and input/output devices	
Setting	**Description**
0,1	RS-232-C serial port 1
2	RS-232-C serial port 2
4	Memory card interface
5	Data server interface

	#7	#6	#5	#4	#3	#2	#1	#0
0100					NCR		CTV	

[Input type] Setting input
[Data type] Bit

 # 1 CTV Character counting for TV check in the comment section of a program.
0: Performed
1: Not performed

 # 3 NCR Output of the end of block (EOB) in ISO code
0: LF, CR, CR are output.
1: Only LF is output.

	#7	#6	#5	#4	#3	#2	#1	#0
0138	MNC							

[Input type] Parameter input
[Data type] Bit

 # 7 MNC DNC operation from the memory card and external device subprogram call from the memory card are:
0: Not performed.
1: Performed.

0983	Path control type of each path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
Set the path control type of each path.
The following two path control types are available:
T series (lathe system) : 0
M series (machining system) : 1

	#7	#6	#5	#4	#3	#2	#1	#0
0984								LCP

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 LCP Set whether the path is a loader control path.
0: The path is not a loader control path.
1: The path is a loader control path.

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 INM Least command increment on the linear axis
0: In mm (metric system machine)
1: In inches (inch system machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1002	IDG			XIK	AZR			JAX

[Input type] Parameter input
[Data type] Bit path

 # 0 JAX Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return
0: 1 axis
1: 3 axes

 # 3 AZR When no reference position is set, the G28 command causes:
0: Reference position return using deceleration dogs (as during manual reference position return) to be executed.
1: Alarm (PS0304) "G28 was specified when no reference position is set" to be displayed.

> **NOTE**
> When reference position return without dogs is specified, (when bit 1 (DLZ) of parameter No.1002 is set to 1) the G28 command specified before a reference position is set causes an alarm PS0304 to be issued, regardless of the setting of AZR.

 # 4 XIK When LRP, bit 1 of parameter No.1401, is set to 0, namely, when positioning is performed using non-linear type positioning, if an interlock is applied to the machine along one of axes in positioning,
0: The machine stops moving along the axis for which the interlock is applied and continues to move along the other axes.
1: The machine stops moving along all the axes.

 # 7 IDG When the reference position is set without dogs, automatic setting of the IDGx parameter (bit 0 of parameter No.1012) to prevent the reference position from being set again is:
0: Not performed.
1: Performed.

> **NOTE**
> When this parameter is set to 0, bit 0 (IDGx) of parameter No. 1012 is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
1004	IPR							

[Input type] Parameter input
[Data type] Bit path

 # 7 IPR When a number with no decimal point is specified, the least input increment of each axis is:
0: Not 10 times greater than the least command increment
1: 10 times greater than the least command increment
When the increment system is IS-A, and bit 0 (DPI) of parameter No. 3401 is set to 1 (fixed-point format), the least input increment cannot be 10 times greater than the least command increment.

	#7	#6	#5	#4	#3	#2	#1	#0
1005			EDMx	EDPx				ZRNx

[Input type] Parameter input
[Data type] Bit axis

 # 0 ZRNx If a move command other than G28 is specified by automatic operation when no reference position return is performed yet after the power is turned on:
0: The alarm (PS0224) "PERFORM REFERENCE POSITION RETURN." is issued.
1: Operation is performed without issuing an alarm.

> **NOTE**
> The state in which a reference position has not been established refers to the following state:
> - When an absolute position detector is not used and reference position return has not been performed even once after power-up
> - When an absolute position detector is used and the association of the machine position with the position detected with the absolute position detector has not been completed (See the description of bit 4 (APZx) of parameter No. 1815.)

 # 4 EDPx In cutting feed, an external deceleration signal in the + direction for each axis is:
0: Invalid
1: Valid

 # 5 EDMx In cutting feed, an external deceleration signal in the - direction for each axis is:
0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx		DIAx		ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

ROTx, ROSx Setting linear or rotation axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No.3624)
0	1	Rotation axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360_. Absolute coordinate values are rounded or not rounded by parameter No.1008#0(ROAx) and #2(RRLx). (3) Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotation axis (B type) (1) Inch/metric conversion, absolute coordinate values and relative coordinate values are not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No.3624) (4) Cannot be used with the rotation axis roll over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

 # 3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

 # 5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007			G90x		RAAx			

[Input type] Parameter input
[Data type] Bit axis

 # 3 RAAx Rotary axis control is:
0: Not exercised.
1: Exercised.
When an absolute command is specified, the rotary axis control function determines the direction of rotation from the sign of the command value and determines an end coordinate from the absolute value of the command value.

> **NOTE**
> RAA is valid when bit 0 (ROA) of parameter No. 1008 is set to 1 and bit 1 (RAB) of parameter No. 1008 is set to 0.
> To use this function, the option for rotary axis control is required.

 # 5 G90x A command for a rotary controlled axis is:
0: Regarded as an absolute/incremental command according to the G90/G91 mode setting.
1: Regarded as an absolute command at all times.

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 ROAx The roll-over function of a rotation axis is
0: Invalid
1: Valid

> **NOTE**
> ROAx specifies the function only for a rotation axis (for which ROTx, #0 of parameter No.1006, is set to 1)

 # 1 RABx In the absolute commands, the axis rotates in the direction
0: In which the distance to the target is shorter.
1: Specified by the sign of command value.

> **NOTE**
> RABx is valid only when ROAx is 1.

 # 2 RRLx Relative coordinates are
0: Not rounded by the amount of the shift per one rotation
1: Rounded by the amount of the shift per one rotation

> **NOTE**
> 1 RRLx is valid only when ROAx is 1.
> 2 Assign the amount of the shift per one rotation in parameter No.1260.

	#7	#6	#5	#4	#3	#2	#1	#0
1013					ISEx	ISDx	ISCx	ISAx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 ISA
 # 1 ISC
 # 2 ISD
 # 3 ISE Increment system of each axis

Increment system	#3 ISE	#2 ISD	#1 ISC	#0 ISA
IS-A	0	0	0	1
IS-B	0	0	0	0
IS-C	0	0	1	0
IS-D	0	1	0	0
IS-E	1	0	0	0

	#7	#6	#5	#4	#3	#2	#1	#0
1015	DWT							

[Input type] Parameter input
[Data type] Bit path

 # 7 DWT When time for dwell per second is specified by P, the increment system:
0: Depends on the increment system
1: Does not depend on the increment system (1 ms)

1020	Program axis name for each axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 67,85 to 90

An axis name (axis name 1: parameter No. 1020) can be arbitrarily selected from 'A', 'B', 'C', 'U', 'V', 'W', 'X', 'Y', and 'Z'. (When G code system A is used with the lathe system, however, 'U', 'V', and 'W' are not selectable.) When bit 0 (EEA) of parameter No. 1000 is set to 1, the length of an axis name can be extended to three characters by setting axis name 2 (parameter No. 1025) and axis name 3 (parameter No. 1026) (extended axis name).
For axis names 2 and 3, a character from '0' to '9' and 'A' to 'Z' of ASCII code can be arbitrarily selected. However, the setting of axis name 3 for each axis is invalid if axis name 2 is not set. Moreover, if a character from '0' to '9' is set as axis name 2, do not use a character from 'A' to 'Z' as axis name 3.

(Tip) ASCII code

Axis name	X	Y	Z	A	B	C	U	V	W
Setting	88	89	90	65	66	67	85	86	87

When G code system A is used with the lathe system, and the character 'X','Y','Z', or 'C' is used as axis name 1 of an axis, a command with 'U','V','W', or 'H' specified for axis name 1 represents an incremental command for the axis.

NOTE
1. When a multiple repetitive canned cycle for turning is used, no character other than 'X','Y', and 'Z' can be used as the address of the axis.
2. When the custom macro function is enabled, the same extended axis name as a reserved word cannot be used. Such an extended axis name is regarded as a reserved word.
3. In a macro call, no extended axis name can be used as an argument.

1022	Setting of each axis in the basic coordinate system

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane) and a three-dimensional tool compensation space (XpYpZp), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotation axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1023	Number of the servo axis for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to Number of controlled axes

Set the servo axis for each control axis.
Usually set to same number as the control axis number.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals

* With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.
  Example)
  When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.
* For tandem controlled axes or electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
  Tandem axis:
  For a master axis, set an odd (1, 3, 5, 7, ...) servo axis number. For a slave axis to be paired, set a value obtained by adding 1 to the value set for the master axis.
  EGB axis:
  For a slave axis, set an odd (1, 3, 5, 7, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

1025	Program axis name 2 for each axis

1026	Program axis name 3 for each axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 48 to 57, 65 to 90

When axis name extension is enabled (when bit 0 (EEA) of parameter No. 1000 is set to 1), the length of an axis name can be extended to a maximum of three characters by setting axis name 2 and axis name 3.

> **NOTE**
> If program axis name 2 is not set, program axis name 3 is invalid.

1031	Reference axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

The unit of some parameters common to all axes such as those for dry run feedrate and single-digit F1 feedrate may vary according to the increment system. An increment system can be selected by a parameter on an axis-by-axis basis. So, the unit of those parameters is to match the increment system of a reference axis. Set which axis to use as a reference axis.

Among the basic three axes, the axis with the finest increment system is generally selected as a reference axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1201					FPC	ZCL		ZPR
					FPC	ZCL		ZPR

[Input type] Parameter input
[Data type] Bit path

 # 0 ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0: Not set automatically
1: Set automatically

> **NOTE**
> ZPR is valid while a workpiece coordinate system function is not provided. If a workpiece coordinate system function is provided, making a manual reference position return always causes the workpiece coordinate system to be established on the basis of the workpiece zero point offset (parameters No. 1220 to No. 1226), irrespective of this parameter setting.

 # 2 ZCL Local coordinate system when the manual reference position return is performed
0: The local coordinate system is not canceled.
1: The local coordinate system is canceled.

> **NOTE**
> ZCL is valid when the workpiece coordinate system option is specified. In order to use the local coordinate system (G52), the workpiece coordinate system option is required.

 # 3 **FPC** When a floating reference position is set with a soft key, the relative position indication is:
0: Not preset to 0 (The relative position indication remains unchanged.)
1: Preset to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
1202						G92		
						G92		

[Input type] Parameter input
[Data type] Bit path

 # 2 **G92** When the CNC has commands G52 to G59 specifying workpiece coordinate systems (optional function), if the G command for setting a coordinate system (G92 for M series, G50 for T series (or the G92 command in G command system B or C)) is specified,
0: G command is executed and no alarm is issued.
1: G command is not executed and an alarm (PS0010) is issued.

1240	Coordinate value of the reference position in the machine coordinate system

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the reference position in the machine coordinate system.

1241	Coordinate value of the second reference position in the machine coordinate system
1242	Coordinate value of the third reference position in the machine coordinate system
1243	Coordinate value of the fourth reference position in the machine coordinate system

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the second to fourth reference positions in the machine coordinate system.

1244	Coordinate value of the floating reference position in the machine coordinate system

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the floating reference position in the machine coordinate system.

1250	Coordinate system of the reference position used when automatic coordinate system setting is performed

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate system of the reference position on each axis to be used for setting a coordinate system automatically.

1260	Amount of a shift per one rotation of a rotation axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

Set the amount of a shift per one rotation of a rotation axis.
For the rotation axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA						NAL	OUT

[Input type] Setting input
[Data type] Bit path

 # 0 OUT The area inside or outside of the stored stroke check 2 is set as an inhibition area
0: Inside
1: Outside

 # 1 NAL When the tool enters the inhibition area of stored stroke limit 1:
0: The overtravel alarm signal is not output.
1: The overtravel alarm signal is output, and the tool is decelerated to a stop.
If manual operation is in progress at this time, the alarm is not output.

> **NOTE**
> When this parameter is set to 1, the alarm is issued if the tool enters stored stroke limit 1 during automatic operation.

 # 7 BFA When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the inter-path interference check function (T series), or a chuck/tail stock barrier (T series) alarm is issued:
0: The tool stops after entering the prohibited area.
1: The tool stops before the prohibited area.

	#7	#6	#5	#4	#3	#2	#1	#0
1301		OTS				NPC		

[Input type] Setting input
[Data type] Bit path

 # 2 NPC As part of the stroke limit check performed before movement, the movement specified in G31 (skip) and G37 (automatic tool length measurement) blocks is:
0: Checked
1: Not checked

> **NOTE**
> This parameter is valid only when the option for stroke check before movement is selected.

 # 6 OTS When the overtravel alarm is issued:
0: The overtravel alarm signal is not output to the PMC.
1: The overtravel alarm signal is output to the PMC.

1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis

1321	Coordinate value I of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.

> **NOTE**
> 1 Specify diameter values for any axes for which diameter programming is specified.
> 2 The area outside the area set by parameter No. 1320 and No. 1321 is a prohibited area.

1322	Coordinate value I of stored stroke check 2 in the positive direction on each axis

1323	Coordinate value I of stored stroke check 2 in the negative direction on each axis

[Input type]	Setting input
[Data type]	Real axis
[Unit of data]	mm, inch, degree (machine unit)
[Minimum unit of data]	Depend on the increment system of the applied axis
[Valid data range]	9 digit of minimum unit of data (refer to standard parameter setting table (A)) (When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate value of stored stroke check 2 on each axis in the + or - direction in the machine coordinate system.

> **NOTE**
> 1 Specify diameter values for any axes for which diameter programming is specified.
> 2 Whether the inside area or outside area is a prohibited area is set using bit 0 (OUT) of parameter No. 1300.

1324	Coordinate value I of stored stroke check 3 in the positive direction on each axis

1325	Coordinate value I of stored stroke check 3 in the negative direction on each axis

[Input type]	Setting input
[Data type]	Real axis
[Unit of data]	mm, inch, degree (machine unit)
[Minimum unit of data]	Depend on the increment system of the applied axis
[Valid data range]	9 digit of minimum unit of data (refer to standard parameter setting table (A)) (When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate value of stored stroke check 3 on each axis in the + or - direction in the machine coordinate system.

> **NOTE**
> 1 Specify diameter values for any axes for which diameter programming is specified.
> 2 The area inside the area set by parameter No. 1324 and No. 1325 is a prohibited area.

1326	Coordinate value II of stored stroke check 1 in the negative direction on each axis

1327	Coordinate value II of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.

When the stored stroke check switch signal EXLM is set to 1, or the stored stroke check switch signal for each axis direction +EXLx is set to 1, parameter No. 1326 and No. 1327 are used for stroke check instead of parameter No.1320 and No. 1321.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The area outside the area set by parameter No. 1326 and No. 1327 is a prohibited area.
3 The EXLM signal is valid only when bit 2 (LMS) of parameter No. 1300 is set to 1.
4 The +EXLx signal is valid only when bit 0 (DLM) of parameter No. 1301 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0			LRP	RPD

[Input type] Parameter input
[Data type] Bit path

 # 0 RPD Manual rapid traverse during the period from power-on time to the completion of the reference position return.
0: Disabled (Jog feed is performed.)
1: Enabled

 # 1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.

When using three-dimensional coordinate conversion, set this parameter to 1.

 # 4 RF0 When cutting feedrate override is 0% during rapid traverse,
0: The machine tool does not stop moving.
1: The machine tool stops moving.

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				NPC

[Input type] Parameter input
[Data type] Bit path

 # 0 NPC Feed per revolution without the position coder (function for converting feed per revolution F to feed per minute F in the feed per revolution mode (G95)) is:
0: Not used
1: Used

 # 4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per rotation.

> **NOTE**
> 1 Specify a feedrate in parameter No.1423.
> 2 For the machining center system, the option for threading/synchronous feed is required.

	#7	#6	#5	#4	#3	#2	#1	#0
1403			HTG					
			HTG					

[Input type] Parameter input
[Data type] Bit path

 # 5 HTG The feedrate for helical interpolation/helical involute interpolation/three-dimensional circular interpolation is:
0: Specified using the feedrate along the tangent to an arc/involute curve/three-dimensional arc
1: Specified using the feedrate along axes including a linear axis (specified axes other than the circular interpolation axis in the case of three-dimensional circular interpolation)

	#7	#6	#5	#4	#3	#2	#1	#0
1404						FM3		

[Input type] Parameter input
[Data type] Bit path

 # 2 FM3 The increment system of an F command without a decimal point in feed per minute is:
0: 1 mm/min (0.01 inch/min for inch input)
1: 0.001 mm/min (0.00001 inch/min for inch input)

	#7	#6	#5	#4	#3	#2	#1	#0
1405								
							FR3	

[Input type] Parameter input
[Data type] Bit path

 # 1 FR3 The increment system of an F command without a decimal point in feed per revolution is:
0: 0.01 mm/rev (0.0001 inch/rev for inch input)
1: 0.001 mm/rev (0.00001 inch/rev for inch input)

1410	Dry run rate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Set the dry run rate at the 100% position on the jog feedrate specification dial. The unit of data depends on the increment system of the reference axis.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1421	F0 rate of rapid traverse override for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Set the F0 rate of the rapid traverse override for each axis.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

(1) When JRV, bit 4 of parameter No.1402, is set to 0 (feed per minute), specify a jog feedrate (feed per minute) under an override of 100%.
(2) When JRV, bit 4 of parameter No.1402, is set to 1 (feed per revolution), specify a jog feedrate (feed per revolution) under an override of 100%.

> **NOTE**
> This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

> **NOTE**
> 1 If 0 is set, the rate set in parameter 1420 (rapid traverse rate for each axis) is assumed.
> 2 When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

1425	FL rate of the reference position return for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

1427	External deceleration rate of rapid traverse for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

Set the external deceleration rate of rapid traverse for each axis.

1428	Reference position return feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

This parameter sets a rapid traverse rate for reference position return operation using deceleration dogs, or for reference position return operation before a reference position is set.

This parameter is also used to set a feedrate for the rapid traverse command (G00) in automatic operation before a reference position is set.

NOTE

1 To this feedrate setting (100%), a rapid traverse override (F0, 25, 50, or 100%) is applicable.
2 For automatic return after completion of reference position return and machine coordinate system establishment, the normal rapid traverse rate is used.
3 As a manual rapid traverse rate before machine coordinate system establishment by reference position return, the jog feedrate or manual rapid traverse rate can be selected with bit 0 (RPD) of parameter No. 1401.

	Before coordinate system establishment	**After coordinate system establishment**
Automatic reference position return (G28)	No.1428	No.1420
Automatic rapid traverse (G00)	No.1428	No.1420
Manual reference position return *1	No.1428	No.1428 *3
Manual rapid traverse	No.1423 *2	No.1424

4 When parameter No. 1428 is set to 0, the following parameter-set feedrates are applied.

	Before coordinate system establishment	**After coordinate system establishment**
Automatic reference position return (G28)	No.1420	No.1420
Automatic rapid traverse (G00)	No.1420	No.1420
Manual reference position return *1	No.1424	No.1424 *3
Manual rapid traverse	No.1423 *2	No.1424

1420: rapid traverse rate
1423: Jog feedrate
1424: Manual rapid traverse rate
*1 : By using bit 2 (JZR) of parameter No. 1401, the jog feedrate can be used for manual reference position return at all times.
*2 : When bit 0 (RPD) of parameter No. 1401 is set to 1, the setting of parameter No. 1424 is used.
*3 : When rapid traverse is used for reference position return without dogs or manual reference position return after reference position establishment, regardless of the deceleration dog, the feedrate for manual reference position return based on these functions is used (the setting of bit 1 (DLF) of parameter No. 1404 is followed).

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Specify the maximum cutting feedrate for each axis.

1432	Maximum cutting feedrate for all axes in the acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Set a maximum cutting feedrate for each axis in the acceleration/deceleration before interpolation mode such as AI contour control. When the acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No. 1430 is used.

1434	Maximum manual handle feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Set a maximum manual handle feedrate for each axis.

1441	External deceleration rate setting 2 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Set external deceleration rate 2 for each axis in rapid traverse.

1444	External deceleration rate setting 3 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
Set external deceleration rate 3 for each axis in rapid traverse.

1450	Change in feedrate per graduation for the manual pulse generator during one-digit F code feed

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 127

Set the constant that determines the change in feedrate as the manual pulse generator is rotated one graduation during one-digit F code feed.

ΔF=Fmaxi/100n (where, i=1 or 2)

In the above equation, n is, the number of revolutions of the manual pulse generator, required to reach feedrate Fmaxi. Fmaxi refers to the upper limit of the feedrate for an one-digit F code feed command, and set it in parameter Nos. 1460 or 1461.

Fmax1: Upper limit of the feedrate for F1 to F4 (parameter No. 1460)
Fmax2: Upper limit of the feedrate for F5 to F9 (parameter No. 1461)

1451	Feedrate for F1

to

1459	Feedrate for F9

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 - +240000.0)

Set Feedrates for one-digit F code feed commands F1 to F9.

When an one-digit F code feed command is executed, as the feedrate is changed by turning the manual pulse generator, these parameter values also change accordingly.

1460	
	Upper feedrate limit for F1 to F4

1461	
	Upper feedrate limit for F5 to F9

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min(machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 - +240000.0)

Set the upper limit of feedrate for the one-digit F code feed command.
As the feedrate increases by turning the manual pulse generator, the feedrate is clamped when it reaches the upper limit set. If an one-digit F code feed command F1 to F4 is executed, the upper limit is that set in parameter 1460. If an one-digit F code feed command F5 to F9 is executed, the upper limit is that set in parameter 1461.

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Input type] Parameter input
[Data type] Bit path

 # 5 NCI An in-position check:
0: Confirms that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time and that the machine position has reached a specified position (the servo positional deviation is within the in-position width set by parameter No. 1827).
1: Confirms only that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time.

	#7	#6	#5	#4	#3	#2	#1	#0
1604								SHP

[Input type] Parameter input
[Data type] Bit path

 # 0 SHP When automatic operation is started, the state equivalent to the specification of G5.1Q1 for AI contour control is:
0: Not set
1: Set
Upon reset, the state where G5.1Q1 is specified is set.

	#7	#6	#5	#4	#3	#2	#1	#0
1606								MNJx

[Input type] Parameter input
[Data type] Bit axis

 # 0 MNJx In manual handle interrupt or automatic manual simultaneous operation (interrupt type):
0: Only cutting feed acceleration/deceleration is enabled, and jog feed acceleration/deceleration is disabled.
1: Both cutting feed acceleration/deceleration and jog feed acceleration/deceleration are applied.

	#7	#6	#5	#4	#3	#2	#1	#0
1610							CTBx	CTLx

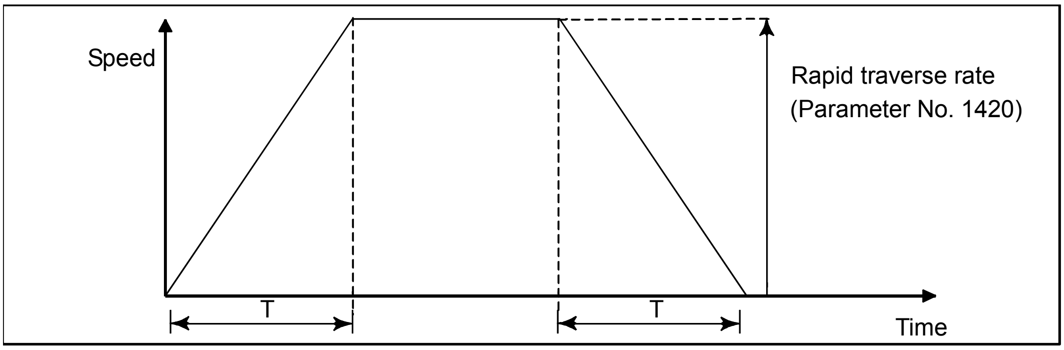
[Input type] Parameter input
[Data type] Bit axis

 # 0 CTLx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

 # 1 CTBx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied.
(depending on the setting in CTLx, bit 0 of parameter No.1610)
1: Bell-shaped acceleration/deceleration is applied.

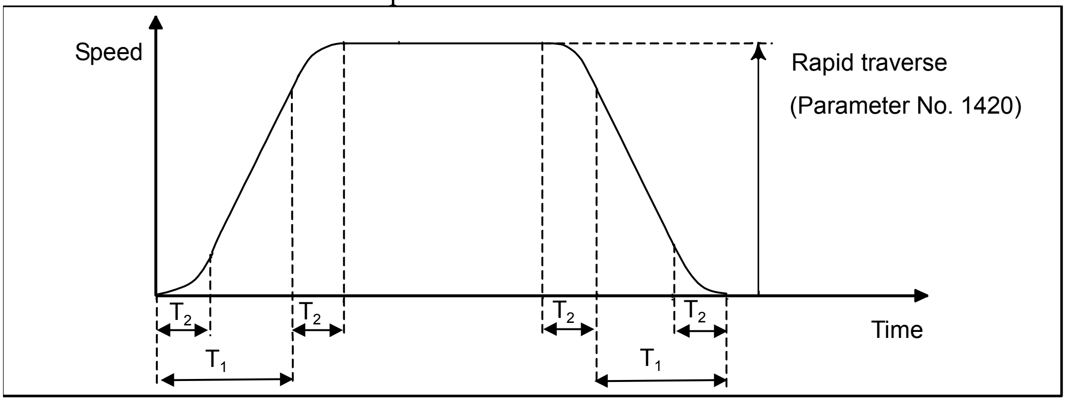
1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Specify a time constant used for acceleration/deceleration in rapid traverse.
[Example]
For linear acceleration/deceleration



T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration



T_1 : Setting of parameter No. 1620

T_2 : Setting of parameter No. 1621 (However, $T_1 \geq T_2$ must be satisfied.)

Total acceleration (deceleration) time : $T_1 + T_2$

Time for linear portion : $T_1 - T_2$

Time for curve portion : $T_2 \times 2$

1622	Time constant of acceleration/deceleration in cutting feed for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed, bell-shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in cutting feed for each axis. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1624	Time constant of acceleration/deceleration in jog feed for each axis.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for acceleration/deceleration in jog feed for each axis.

1660	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration rate in acceleration/ deceleration before interpolation for each axis.
If a value greater than 100000.0 is set, the value is clamped to 100000.0.
If 0 is set, the specification of 100000.0 is assumed. If 0 is set for all axes, however, acceleration/deceleration before interpolation is not performed.

If a maximum allowable acceleration rate set for one axis is greater than a maximum allowable acceleration rate set for another axis by a factor or 2 or more, the feedrate at a corner where the direction of travel abruptly changes can decrease temporarily.

1671	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for linear rapid traverse for each axis, or maximum allowable reference acceleration rate in optimum torque acceleration/deceleration

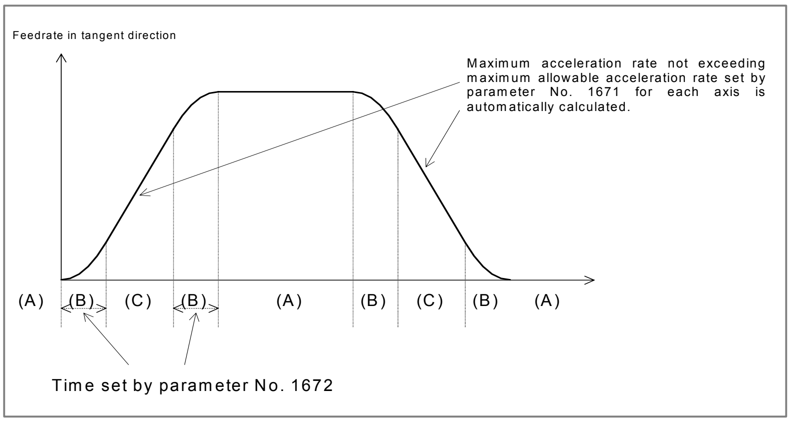
[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

(1) Set a maximum allowable acceleration rate in acceleration/ deceleration before interpolation for linear rapid traverse.
If a value greater than 100000.0, the value is clamped to 100000.0.
If 0 is set, the specification of the following is assumed:
1000.0 mm/sec/sec
100.0 inch/sec/sec
100.0 degrees/sec/sec
If 0 is specified for all axes, however, acceleration/deceleration before interpolation is not performed.
(2) Maximum allowable reference acceleration rate in optimum torque acceleration/deceleration

1672	**Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse, or acceleration change time of bell-shaped acceleration/deceleration in optimum torque acceleration/deceleration**

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

(1) Set an acceleration change time of bell-shaped acceleration/ deceleration for linear rapid traverse (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1671: time of (B) in the figure below).
(2) Set an acceleration change time of bell-shaped acceleration/ deceleration in optimum torque acceleration/deceleration (time for changing from the state of constant feedrate (A) to the state of acceleration/deceleration (C) at the acceleration rate calculated from optimum torque acceleration/deceleration: time of (B) in the figure below).



1710	Minimum deceleration ratio (MDR) for inner circular cutting feedrate change by automatic corner override

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100

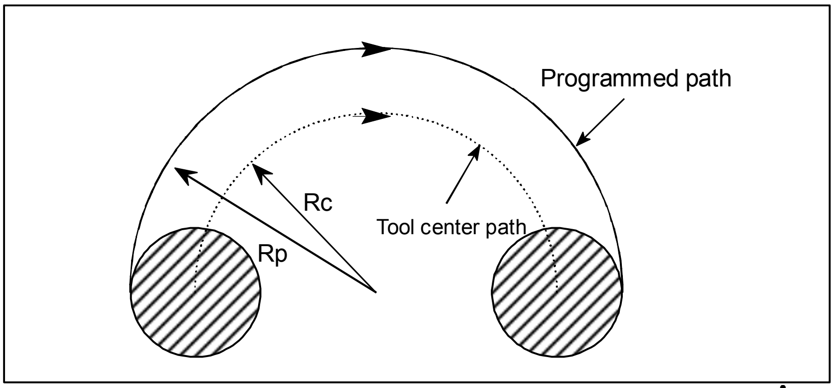
Set a minimum deceleration ratio (MDR) for an inner circular cutting feedrate change by automatic corner override.

In the case of circular cutting offset inward, the actual feedrate is determined by a specified feedrate (F) as follows:

$$F \times \frac{Rc}{Rp}$$

(Rc:Radius of tool center path
Rp:Programmed radius)

Thus, the feedrate along the programmed path satisfies the specified value of F.



However, if Rc is too small when compared with Rp, Rc/Rp $\fallingdotseq$ 0 results to stop the tool. So, a minimum deceleration ratio (MDR) is set, and the feedrate of the tool is set to F×(MDR) when Rc/Rp ≤ MDR.

1711	Inner determination angle (θp) for inner corner override

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 2 to 178

Set an inner determination angle for inner corner override in automatic corner overriding.

1712	Override value for inner corner override

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 1 to 100

Set an inner corner override value in automatic corner overriding.

1713	Start distance (Le) for inner corner override

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

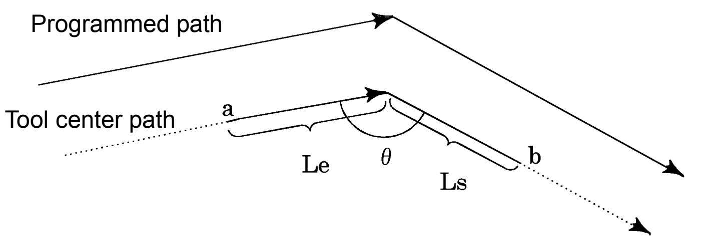
Set a start distance for inner corner override in automatic corner overriding.

1714	End distance (Ls) for inner corner override

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set an end distance for inner corner override in automatic corner overriding.

When $\theta \leq \theta p$, an inner corner is assumed. (Parameter No. 1711 is used to set θp.)
When a corner is determined to be an inner corner, an override is applied to the feedrate in the range of Le in the previous block from the intersection of the corner and in the range of Ls in the next block from the intersection of the corner.
Distances Le and Ls represent linear distances from the intersection of a corner to points on the tool center path.
Le and Ls are set in parameter No. 1713 and No. 1714.



An override is applied to the range from point a to point b.

1732	**Minimum allowable feedrate for the deceleration function based on acceleration in circular interpolation**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

With the deceleration function based on acceleration in circular interpolation, an optimum feedrate is automatically calculated so that acceleration produced by changing the move direction in circular interpolation does not exceed the maximum allowable acceleration rate specified in parameter No. 1735.

If the radius of an arc is very small, a calculated feedrate may become too low.

In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

> **NOTE**
> During involute interpolation, the minimum allowable feedrate of "clamping of acceleration near a basic circle" in involute interpolation automatic feedrate control is used.

1735	**Maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation.

Feedrate is controlled so that acceleration produced by changing the move direction in circular interpolation does not exceed the value specified in this parameter.

For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled.

If a different value is set in this parameter for each axis, a feedrate is determined from the smaller of the acceleration rates specified for the two circular axes.

> **NOTE**
> During involute interpolation, the minimum allowable feedrate of "clamping of acceleration near a basic circle" in involute interpolation automatic feedrate control is used.

1737	Maximum allowable acceleration rate for the deceleration function based on acceleration in AI contour control for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration rate produced by changing the tool move direction.

For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled. If 0 is set for all axes, the deceleration function based on acceleration is not performed.

In circular interpolation, however, the deceleration function based on feedrate control using acceleration in circular interpolation (parameter No. 1735) is enabled.

1738	Minimum allowable feedrate for the deceleration function based on acceleration in AI contour control

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

With the deceleration function based on acceleration in AI contour control, a feedrate most suitable for a desired figure is automatically calculated.

Depending on the figure, however, the calculated feedrate may become too low.

In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

If overriding using the deceleration function based on cutting load is enabled, a feedrate lower than the minimum allowable feedrate may be used.

1769	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

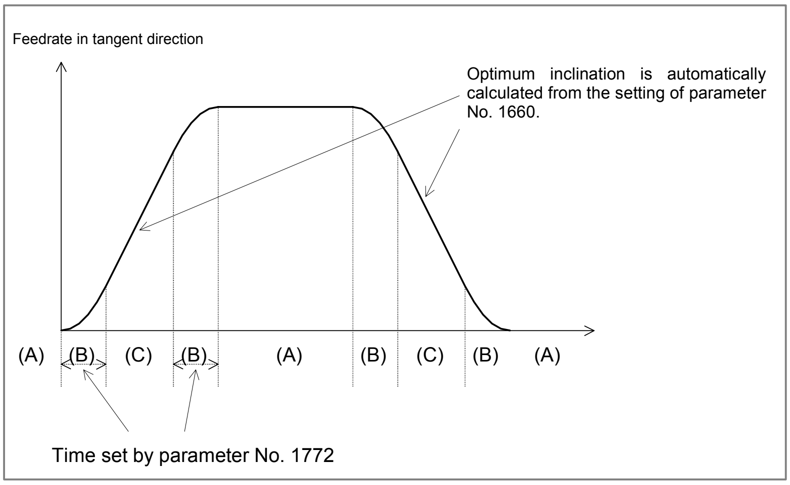
In the acceleration/deceleration before interpolation mode as in AI contour control, not the ordinary time constant (parameter No. 1622) but the value of this parameter is used.

Be sure to specify the same time constant value for all axes except for a special application. If different values are set, correct linear and circular figures cannot be obtained.

1772	Acceleration change time of bell-shaped acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

Set an acceleration change time of bell-shaped acceleration/ deceleration before interpolation (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1660: time of (B) in the figure below).



1783	Maximum allowable feedrate difference for feedrate determination based on corner feedrate difference

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

If a feedrate component change for each axis exceeding the value set in this parameter occurs at the joint of blocks, the feedrate determination function based on corner feedrate difference finds a feedrate not exceeding the set value and performs deceleration by using acceleration/deceleration before interpolation. Thus, a shock to the machine and machining error at a corner can be reduced.

1788	Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration change rate for each axis in feedrate control based on acceleration change under control on the rate of change of acceleration.

For an axis with 0 set in this parameter, feedrate control based on acceleration change is disabled.

If 0 is set for all axes, feedrate control based on acceleration change is not exercised.

1789	Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis (linear interpolation)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration change rate for each axis in feedrate control based on acceleration change under control on the rate of change of acceleration in successive linear interpolation operations.

In feedrate control based on acceleration change at a corner between linear interpolation operations, the maximum allowable acceleration change rate not set in parameter No. 1788 but set in this parameter is valid.

For an axis with 0 set in this parameter, the maximum allowable acceleration change rate set in parameter No. 1788 is valid.
Feedrate control based on acceleration change is disabled for an axis with 0 set in parameter No. 1788, so that the setting of this parameter for such an axis is ignored.

1790	Ratio of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 50

Set the ratio of the change time of the rate of change of acceleration to the change time of acceleration(*1) by percentage (%) in smooth bell-shaped acceleration/deceleration before look-ahead interpolation.
If 0 is set in this parameter or a value not within the valid data range is specified in this parameter, smooth bell-shaped acceleration/ deceleration before look-ahead interpolation is not performed.

(*1)
Parameter No. 1772 for acceleration/deceleration before look-ahead interpolation (cutting feed).
Parameter No. 1672 for acceleration/deceleration before interpolation in linear rapid traverse, or for optimum torque acceleration/ deceleration.

	#7	#6	#5	#4	#3	#2	#1	#0
1802						DC2x	DC4x	

[Input type] Parameter input
[Data type] Bit axis

 # 1 DC4x When the reference position is established on the linear scale with reference marks:
0: An absolute position is established by detecting three reference marks.
1: An absolute position is established by detecting four reference marks.

 # 2 DC2x Reference position establishment operation for a linear scale with reference marks is performed as follows:
0: The setting of bit 1 (DC4) of parameter No. 1802 is followed.
1: An absolute position is established by detecting two reference marks.

NOTE

1 When this parameter is set to 1, specify the direction of the scale zero point by setting bit 4 (SCP) of parameter No. 1817.

2 When a rotary encoder with absolute address reference marks is used, this parameter is invalid. Even when this parameter is set to 1, the setting of bit 1 (DC4) of parameter No. 1802 is followed.

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx	DCRx		OPTx	

[Input type] Parameter input
[Data type] Bit axis

NOTE

When this parameter is set, the power must be turned off before operation is continued.

 # 1 OPTx Position detector

0: A separate pulse coder is not used.
1: A separate pulse coder is used.

NOTE

Set this parameter to 1 when using a linear scale with reference marks or a linear scale with an absolute address zero point (full-closed system).

 # 3 DCRx As a scale with absolute address reference marks:

0: A rotary encoder with absolute address reference marks is not used.
1: A rotary encoder with absolute address reference marks is used.

NOTE

When using a rotary encoder with absolute address reference marks, set also bit 2 (DCLx) of parameter No. 1815 to 1.

 # 4 APZx Machine position and position on absolute position detector when the absolute position detector is used

0: Not corresponding
1: Corresponding

When an absolute position detector is used, after primary adjustment is performed or after the absolute position detector is replaced, this parameter must be set to 0, power must be turned off and on, then manual reference position return must be performed. This completes the positional correspondence between the machine position and the position on the absolute position detector, and sets this parameter to 1 automatically.

 # 5 **APCx** Position detector
0: Other than absolute position detector
1: Absolute position detector (absolute pulse coder)

	#7	#6	#5	#4	#3	#2	#1	#0
1817		TANx						

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 6 **TANx** Tandem control
0: Not used
1: Used

> **NOTE**
> Set this parameter to both master axis and slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1818					SDC		RF2x	RFSx

[Input type] Parameter input
[Data type] Bit axis

 # 0 **RFSx** If G28 is specified for an axis for which a reference position is not established (ZRF = 0) when a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used:
0: A movement is made to the reference position after reference position establishment operation.
1: No movement is made after reference position establishment operation, but the operation is completed.

> **NOTE**
> This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

 # 1 **RF2x** If G28 is specified for an axis for which a reference position is already established (ZRF = 1) when a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used:
0: A movement is made to the reference position.
1: No movement is made to the intermediate position and reference position, but the operation is completed.

> **NOTE**
> This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

 # 3 SDCx A linear scale with an absolute address zero point is:
0: Not used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
1819						DATx		

[Input type] Parameter input
[Data type] Bit axis

 # 2 DATx When a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used, the automatic setting of parameter No. 1883 and No. 1884 at manual reference position return time is:
0: Not performed.
1: Performed.

1820	Command multiplier for each axis (CMR)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] See below :
Set a command multiplier indicating the ratio of the least command increment to the detection unit for each axis.
Least command increment = detection unit × command multiplier

Relationship between the increment system and the least command increment

(1) T series

			Least input increment	Least command increment
IS-B	Millimeter machine	Millimeter input	0.001 mm (diameter specification)	0.0005 mm
			0.001 mm (radius specification)	0.001 mm
		Inch input	0.0001 inch (diameter specification)	0.0005 mm
			0.0001 inch (radius specification)	0.001 mm
	Inch machine	Millimeter input	0.001 mm (diameter specification)	0.00005 inch
			0.001 mm (radius specification)	0.0001 inch
		Inch input	0.0001 inch (diameter specification)	0.00005 inch
			0.0001 inch (radius specification)	0.0001 inch
	Rotation axis		0.001 deg	0.001 deg

			Least input increment	Least command increment
IS-C	Millimeter machine	Millimeter input	0.0001 mm (diameter specification)	0.00005 mm
			0.0001 mm (radius specification)	0.0001 mm
		Inch input	0.00001 inch (diameter specification)	0.00005 mm
			0.00001 inch (radius specification)	0.0001 mm
	Inch machine	Millimeter input	0.0001 mm (diameter specification)	0.000005 inch
			0.0001 mm (radius specification)	0.00001 inch
		Inch input	0.00001 inch (diameter specification)	0.000005 inch
			0.00001 inch (radius specification)	0.00001 inch
	Rotation axis		0.0001 deg	0.0001 deg

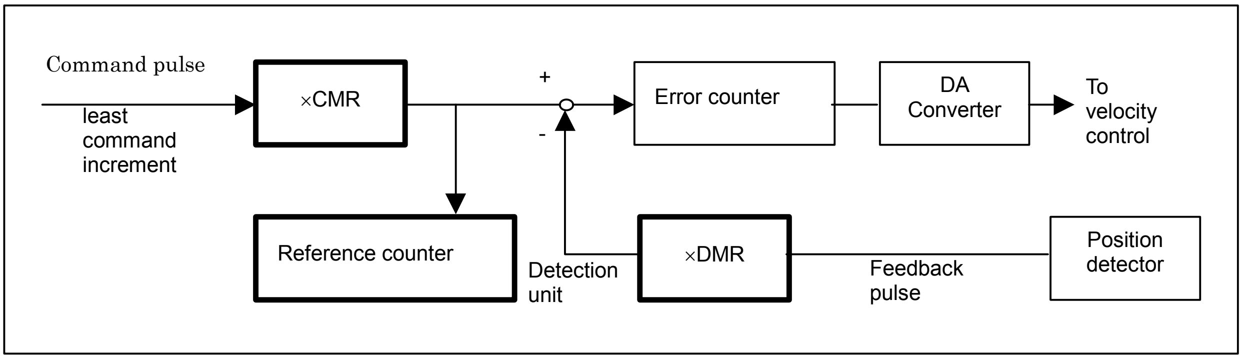
			Least input increment	Least command increment
IS-D	Millimeter machine	Millimeter input	0.00001 mm (diameter specification)	0.000005 mm
			0.00001 mm (radius specification)	0.00001 mm
		Inch input	0.000001 inch (diameter specification)	0.000005 mm
			0.000001 inch (radius specification)	0.00001 mm
	Inch machine	Millimeter input	0.00001 mm (diameter specification)	0.0000005 inch
			0.00001 mm (radius specification)	0.000001 inch
		Inch input	0.000001 inch (diameter specification)	0.0000005 inch
			0.000001 inch (radius specification)	0.000001 inch
	Rotation axis		0.00001 deg	0.00001 deg

			Least input increment	Least command increment
IS-E	Millimeter machine	Millimeter input	0.000001 mm (diameter specification)	0.0000005 mm
			0.000001 mm (radius specification)	0.000001 mm
		Inch input	0.0000001 inch (diameter specification)	0.0000005 mm
			0.0000001 inch (radius specification)	0.000001 mm
	Inch machine	Millimeter input	0.000001 mm (diameter specification)	0.00000005 inch
			0.000001 mm (radius specification)	0.0000001 inch
		Inch input	0.0000001 inch (diameter specification)	0.00000005 inch
			0.0000001 inch (radius specification)	0.0000001 inch
	Rotation axis		0.000001 deg	0.000001 deg

(2) M series

Increment system	Least input increment and least command increment					
	IS-A	IS-B	IS-C	IS-D	IS-E	Unit
Millimeter machine	0.01	0.001	0.0001	0.00001	0.000001	mm
Millimeter input	0.001	0.0001	0.00001	0.000001	0.0000001	inch
Rotation axis	0.01	0.001	0.0001	0.00001	0.000001	deg

Setting command multiply (CMR), detection multiply (DMR), and the capacity of the reference counter



Set CMR and DMR so that the pulse weight of + input (command from the CNC) into the error counter matches the pulse weight of -input (feedback from the position detector).
[Least command increment]/CMR=[Detection unit]=[Feedback pulse unit]/DMR
[Least command increment]: Minimum unit of commands issued from the CNC to the machine
[Detection unit]: Minimum unit for machine position detection

The unit of feedback pulses varies, depending on the type of detector.
[Feedback pulse unit]=[Amount of travel per rotation of the pulse coder]/[Number of pulses per rotation of the pulse coder]

As the size of the reference counter, specify the grid interval for the reference position return in the grid method.
[Size of the reference counter]=[Grid interval]/[Detection unit]
[Grid interval]=[Amount of travel per rotation of the pulse coder]

The setting of a command multiplier is as follows:
(1) When command multiplier is 1 to 1/27
Set value = 1 / command multiplier + 100
Valid data range : 101 to 127

(2) When command multiply is 0.5 to 48
Set value = 2 × command multiplier
Valid data range : 1 to 96

> **NOTE**
> If a feedrate exceeding the feedrate found by the expression below is used, an incorrect travel amount may result or a servo alarm may be issued. Be sure to use a feedrate not exceeding the feedrate found by the following expression:
> Fmax[mm/min] = $196602 \times 10^4 \times$ least command increment / CMR

1821	Reference counter size for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.
When a linear scale with absolute address reference marks is used, set the interval of mark 1.

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.
If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm (SV0411) is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter.

1829	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in the stopped state for each axis.
If, in the stopped state, the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm (SV0410) is generated, and operation is stopped immediately (as in emergency stop).

1838	Position deviarion limit for each axis in moving during safety check

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Paramete input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Position deviation limit for each axis during moving for safety check of Dual Check Safety function is specified.
If position deviation of a moving axis exceeds position deviation limit while Safety Check is carried out (Safety Monitoring Request "*VLDVx" =0), a servo alarm (SV0475, SV1071) is generated and axes are stopped immediately like emergency stop state..
In Dual Check Safety function, positin deviation is always checked by CNC and Servo. In case that Safety Check is carried out (Safety Monitoring Request "*VLDVx" =0), the servo alarm (SV0475,SV1071) is generated when each CPU finds out that the deviation exceeds position deviation limit in moving state.

1841	Position deviation limit of each axis in moving state during other than Dual Check Safety monitoring (for Dual Check Safety Function)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2 word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in moving state for each axis for Dual Check Safety function, in case that Safety Check is not carried out (Safety Monitoring Request "*VLDVx"=1).
In case that Safety Check is not carried out (Safety Monitoring Request "*VLDVx" =1), servo alarm (SV0475,SV1071) is generated and operation is stopped immediately (as in emergency stop), when

each CPU finds out that the deviation exceeds position deviation limit in moving state.
If the value of this parameter is “0”, the parameter No.1828 is used for the value of deviation limit in moving state.

In case that Safety Check is carried out (Safety Monitoring Request “*VLDVx” =0), the parameter No.1838 is used for the value of deviation limit in moving state.

1851	Backlash compensating value for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the backlash compensating value for each axis.
When the machine moves in a direction opposite to the reference position return direction after the power is turned on, the first backlash compensation is performed.

1882	Interval of mark 2 of a linear scale with absolute address reference marks

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set the interval of mark 2 of a linear scale with absolute address reference marks.

1883	Distance 1 from the scale zero point to reference position

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

1884	Distance 2 from the scale zero point to reference position

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999 to 999

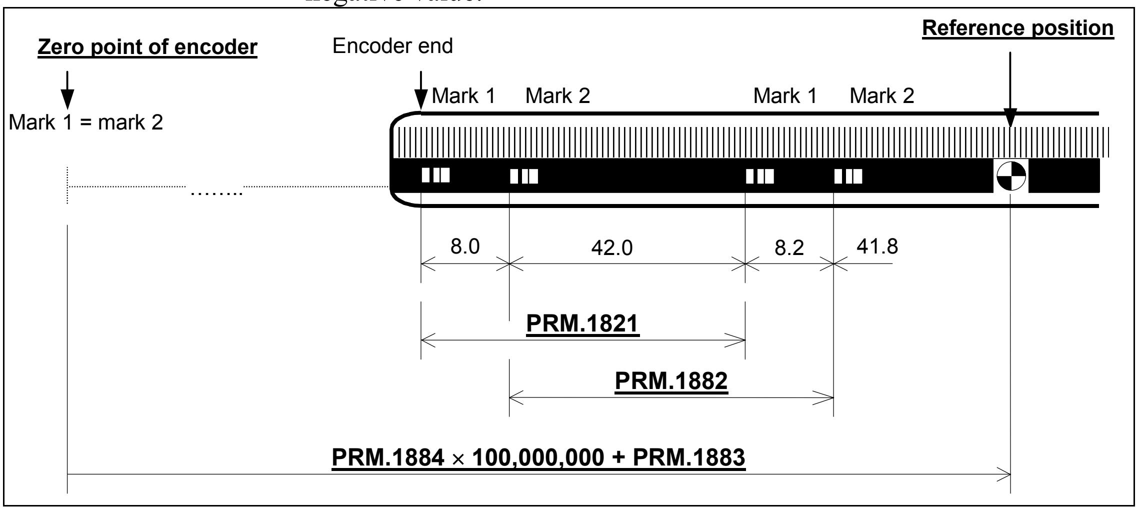
Use this parameter when the distance from the scale zero point to the reference position exceeds the setting range specified in parameter No. 1883.

Parameter No. 1883 and No. 1884 are used to set the distance from the scale zero point to the reference position on a linear scale with absolute address reference marks or a linear scale with an absolute address zero point.

Distance from the zero point to the reference position of a linear scale
= No. 1884 × 1,000,000,000 + No. 1883

The scale zero point represents a point where mark 1 and mark 2 match. Usually, this point is a virtual point that does not physically exist on the scale. (See the figure below.)

If the reference position is placed in the + direction when viewed from the scale zero point, set a positive value. If the reference position is placed in the - direction when viewed from the scale zero point, set a negative value.



	#7	#6	#5	#4	#3	#2	#1	#0
1902							ASE	FMD

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

 # 0 FMD The FSSB setting mode is:
0: Automatic setting mode.
(When the relationship between an axis and amplifier is defined on the FSSB setting screen, parameter Nos. 1023, 1905, 1936 to 1939, and 14340 to 14407 (plus parameter Nos. 14408 to 14425 and 14444 to 14459 if an additional axis board is attached) are automatically set.
1: Manual setting 2 mode.
(Parameter Nos. 1023, 1905, 1936 to 1939 and 14340 to 14407 (plus parameter Nos. 14408 to 14425 and 14444 to 14459 if an additional axis board is attached) are to be manually set.)

 # 1 ASE When automatic setting mode is selected for FSSB setting (when the FMD parameter (bit 0 of parameter No.1902) is set to 0), automatic setting is:
0: Not completed.
1: Completed.
This bit is automatically set to 1 upon the completion of automatic setting.

	#7	#6	#5	#4	#3	#2	#1	#0
1905	PM2	PM1				PM4	PM3	

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

 # 1 PM3 The third separate detector interface unit is:
0: Not used.
1: Used.

 # 2 PM4 The fourth separate detector interface unit is:
0: Not used.
1: Used.

 # 6 PM1 The first separate detector interface unit is:
0: Not used.
1: Used.

 # 7 PM2 The second separate detector interface unit is:
0: Not used.
1: Used.

NOTE
When automatic setting mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 0), this parameter is automatically set when input is performed with the FSSB setting screen. When manual setting 2 mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 1), this parameter must be set directly. When a separate detector interface unit is used, a connector number must be set in the corresponding parameter (No.1936, No.1937, No.1938, or No.1939).

1936	Connector number of the first separate detector interface unit
1937	Connector number of the second separate detector interface unit
1938	Connector number of the third separate detector interface unit
1939	Connector number of the fourth separate detector interface unit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

Set the connector numbers corresponding to connectors to be connected when the separate detector interface unit set by bit 1, 2, 6, or 7 of parameter No. 1905 is used. The values to be set are indicated below.
Within one separate detector interface unit, use connector numbers sequentially. No intermediate number may be omitted.

Correspondence between connectors and connector numbers	
Connector	**Connector number**
JF101	0
JF102	1
JF103	2
JF104	3
JF105	4
JF106	5
JF107	6
JF108	7

Example of setting)

	Separate detector connection destination				Parameter setting				
Controlled axis	Connectors for 1st unit	Connectors for 2nd unit	Connectors for 3rd unit	Connectors for 4th unit	No. 1936	No. 1937	No. 1938	No. 1939	No.1905 (#7,#6,#2,#1)
X1	JF101	-	-	-	0	-	-	-	0,1,0,0
Y1	-	JF102	-	-	-	1	-	-	1,0,0,0
Z1	-	-	JF102	-	-	-	1	-	0,0,0,1
X2	-	JF101	-	-	-	0	-	-	1,0,0,0
Y2	-	-	-	JF101	-	-	-	0	0,0,1,0
Z2	-	-	-	-	-	-	-	-	0,0,0,0
A1	-	-	JF101	-	-	-	0	-	0,0,0,1
B1	-	-	-	JF102	-	-	-	1	0,0,1,0
C1	-	JF104	-	-	-	3	-	-	1,0,0,0
A2	JF102	-	-	-	1	-	-	-	0,1,0,0
B2	-	JF103	-	-	-	2	-	-	1,0,0,0
C2	-	-	-	JF103	-	-	-	2	0,0,1,0

NOTE

When automatic setting mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 0), these parameters are automatically set when input is performed with the FSSB setting screen.
When manual setting 2 mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 1), these parameters must be set directly.

Parameters No.2000 to 2999 are for digital servo, The following parameters are not explained in this manual. Refer to FANUC AC SERVO MOTOR α*i* series PARAMETER MANUAL (B-65270EN)

	#7	#6	#5	#4	#3	#2	#1	#0
2011	XIAx							

[Input type] Parameter input
[Data type] Bit axis

 # 7 XIAx Temporary absolute coordinate setting is:
0: Not used.
1: Used.

NOTE

1 When temporary absolute coordinate setting is used, bit 1 (OPTx) of parameter No. 1815, bit 5 (APCx) of parameter No. 1815, parameter No. 1874, and parameter No. 1875 must be set.
2 The setting of this parameter becomes effective after the power is turned off then back on.

2031	Torque command difference threshold for torque difference alarm

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 14564

If the absolute value of a torque command difference between two axes exceeds the value set in this parameter, an alarm is issued.
Set the same value for the two axes placed under axis synchronous control.
The servo axis number combination of a set of synchronous master and slave axes must be such that an odd number is assigned to the master axis and the immediately following even number is assigned to the slave axis like (1,2) and (3,4).

3012	Skip signal assignment address

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the skip signal (SKIPn) is to be assigned.

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the option configuration of the I/O Link, the actually usable X addresses are:
X0 to X127, X200 to X327, X400 to X527, X600 to X727

3013	X address to which the deceleration signal for reference position return is assigned

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 727

Set an address to which the deceleration signal (*DECn) for reference position return for each axis is to be assigned.

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the option configuration of the I/O Link, the actually usable X addresses are:
X0 to X127, X200 to X327, X400 to X527, X600 to X727

3019	Address to which the PMC axis control skip signal and the measurement position arrival signal are assigned

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the PMC axis control skip signal ESKIP and the measurement position arrival signals (XAE, YAE, and ZAE (M series) or XAE and ZAE (T series)) are to be assigned.

Example 1. When No.3012 is set to 5 and No.3019 is set to 6

When XSG (bit 2 of parameter No. 3008) is 1, the PMC axis control skip signal, and measurement position arrival signal are allocated to X0006 and the skip signal is allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X005	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(T series)
	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X006		ESKIP	-MIT2	+MIT2	-MIT1	+MIT1	ZAE	XAE	(T series)
		ESKIP				ZAE	YAE	XAE	(M series)

Example 2. When No.3012 is set to 5 and No.3019 is set to 5

When XSG (bit 2 of parameter No. 3008) is 1, the PMC axis control skip signal, measurement position arrival signal, and skip signal are allocated to X0005.

<table>
<tr><td></td><td>#7</td><td>#6</td><td>#5</td><td>#4</td><td>#3</td><td>#2</td><td>#1</td><td>#0</td><td></td></tr>
<tr><td rowspan="2">X005</td><td rowspan="2">SKIP</td><td>ESKIP</td><td>-MIT2</td><td>+MIT2</td><td>-MIT1</td><td>+MIT1</td><td>ZAE</td><td>XAE</td><td rowspan="2">(T series)</td></tr>
<tr><td>SKIP6</td><td>SKIP5</td><td>SKIP4</td><td>SKIP3</td><td>SKIP2</td><td>SKIP8</td><td>SKIP7</td></tr>
</table>

<table>
<tr><td>#7</td><td>#6</td><td>#5</td><td>#4</td><td>#3</td><td>#2</td><td>#1</td><td>#0</td><td></td></tr>
<tr><td rowspan="2">SKIP</td><td>ESKIP</td><td rowspan="2">SKIP5</td><td rowspan="2">SKIP4</td><td rowspan="2">SKIP3</td><td>ZAE</td><td>YAE</td><td>XAE</td><td rowspan="2">(M series)</td></tr>
<tr><td>SKIP6</td><td>SKIP2</td><td>SKIP8</td><td>SKIP7</td></tr>
</table>

NOTE

This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the option configuration of the I/O Link, the actually usable X addresses are:
X0 to X127, X200 to X327, X400 to X527, X600 to X727

3021	Address to which an axis signal is assigned

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7, 10 to 17, 20 to 27, ... , 90 to 97

For each axis of the CNC, set a PMC interface address.
Set a value according to the tables below.

Value of parameter No. 3021 (tens digit)

Setting value	Input signal address	Output signal address
0	G0000 to G0999	F0000 to F0999
1	G1000 to G1999	F1000 to F1999
	:	
9	G9000 to G9999	F9000 to F9999

Value of parameter No. 3021 (ones digit)

Setting value	Input signal address	Output signal address
0	#0	#0
1	#1	#1
	:	
7	#7	#7

[Example of setting]

Axis number	No.3021	Signal allocation
1	0	+J1<G0100.0>, -J1<G0102.0>, ZP1<F0090.0>, ...
2	1	+J2<G0100.1>, -J2<G0102.1>, ZP2<F0090.1>, ...
3	2	+J3<G0100.2>, -J3<G0102.2>, ZP3<F0090.2>, ...
4	10	+J4<G1100.0>, -J4<G1102.0>, ZP4<F1090.0>, ...
5	11	+J5<G1100.1>, -J5<G1102.1>, ZP5<F1090.1>, ...

If eight or less axes are used per path, the following signal allocation results when 0 is set for all axes:

Axis 1 of path 1 = Setting equivalent to 0
Axis 2 of path 1 = Setting equivalent to 1
:
Axis 1 of path 2 = Setting equivalent to 10
:

NOTE
Set this parameter when more than eight axes are used per path.
The valid data range varies, depending on the NC system type.

3022 **Address to which a spindle signal is assigned**

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0to3,10to13,20to23, ... ,90to93
For each axis of the CNC, set a PMC interface address.
Set a value according to the tables below.

Value of parameter No. 3022 (tens digit)

Setting value	Input signal address	Output signal address
0	G0000toG0999	F0000toF0999
1	G1000toG1999	F1000toF1999
	:	
9	G9000toG9999	F9000toF9999

Value of parameter No. 3022 (ones digit)

Setting value	Input signal address	Output signal address
0	Bit position A	Bit position A
1	Bit position B	Bit position B
2	Bit position C	Bit position C
3	Bit position D	Bit position D

(The bit positions A, B, C, and D vary, depending on the type of signal.)

[Example of setting]

Spindle number	No.3022	Signal allocation
1	0	TLMLA<G0070.0>, TLMHA<G0070.1>, ALMA<F0045.0>, ...
2	1	TLMLB<G0074.0>, TLMHB<G0074.1>, ALMB<F0049.0>, ...
3	10	TLMLA<G1070.0>, TLMHA<G1070.1>, ALMA<F1045.0>, ...
4	11	TLMLB<G1074.0>, TLMHB<G1074.1>, ALMB<F1049.0>, ...

If four or less axes are used per path, the following signal allocation results when 0 is set for all axes:

Axis 1 of path 1 = Setting equivalent to 0
Axis 2 of path 1 = Setting equivalent to 1
:
Axis 1 of path 2 = Setting equivalent to 10
:

> **NOTE**
> Set this parameter when more than four axes are used per path.
> The valid data range varies, depending on the system software.

3030	Allowable number of digits for the M code

3031	Allowable number of digits for the S code

3032	Allowable number of digits for the T code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8

Set the allowable numbers of digits for the M, S, and T codes.
When 0 is set, the allowable number of digits is assumed to be 8.

3033	Allowable number of digits for the B code (second auxiliary function)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8

Set the allowable number of digits for the second auxiliary function.
When 0 is set, the allowable number of digits is assumed to be 8.
To enable a decimal point to be specified, bit 0 (AUP) of parameter No. 3450 must be set to 1. In this case, the allowable number of digits set in this parameter includes the number of decimal places.
If a value exceeding the allowable number of digits is specified, the alarm (PS0003) is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC		DRC		PPD			MCN
	DAC	DAL	DRC	DRL	PPD			MCN

[Input type] Parameter input
[Data type] Bit path

 # 0 MCN Machine position
0: Regardless of whether input is made in mm or inches, the machine position is displayed in mm for millimeter machines, or in inches for inch machines.
1: When input is made in mm, the machine position is displayed in mm, and when input is made in inches, the machine position is displayed in inches accordingly.

 # 3 PPD Relative position display when a coordinate system is set
0: Not preset
1: Preset

NOTE

If any of the following is executed when PPD is set to 1, the relative position display is preset to the same value as the absolute position display:
(1) Manual reference position return
(2) Coordinate system setting based on G92 (G50 for G code system A on the lathe system)
(3) Workpiece coordinate system presetting based on G92.1 (G50.3 for G code system A on the lath system)
(4) When a T code for the lathe system is specified, the relative position display is preset to the same value as the absolute position display.

 # 4 DRL Relative position
0: The actual position displayed takes into account tool length offset.
1: The programmed position displayed does not take into account tool length offset.

 # 5 DRC When a relative position is displayed:
0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

 # 6 DAL Absolute position
- 0: The actual position displayed takes into account tool length offset.
- 1: The programmed position displayed does not take into account tool length offset.

 # 7 DAC When an absolute position is displayed:
- 0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
- 1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3111		OPS	OPM				SPS	SVS

[Input type] Setting input
[Data type] Bit path

 # 0 SVS Servo setting screen and servo tuning screen
- 0: Not displayed
- 1: Displayed

 # 1 SPS Spindle tuning screen
- 0: Not displayed
- 1: Displayed

 # 5 OPM Operating monitor display is:
- 0: Not provided.
- 1: Provided.

 # 6 OPS The speedometer on the operating monitor screen displays:
- 0: Spindle motor speed.
- 1: Spindle speed.

	#7	#6	#5	#4	#3	#2	#1	#0
3115							NDAx	NDPx

[Input type] Parameter input
[Data type] Bit axis

 # 0 NDPx The current position is:
- 0: Displayed.
- 1: Not displayed.

> **NOTE**
> If using the electric gear box function (EGB), set 1 for the dummy axis of EGB to suppress position display.

 # 1 NDAx The current position and the remaining amount of travel in the absolute coordinate system and relative coordinate system are:
0: Displayed.
1: Not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3129							DAP	DRP

[Input type] Parameter input
[Data type] Bit path

 # 0 DRP For relative coordinate display:
0: The actual position considering a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

 # 1 DAP For absolute coordinate display:
0: The actual position considering a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 65 to 90
In order to distinguish axes under parallel operation, synchronization control, and tandem control, specify a subscript for each axis name.

Setting value	Meaning
0	Each axis is set as an axis other than a parallel axis, synchronization control axis, and tandem control axis.
1 to 9	A set value is used as a subscript.
65 to 90	A set letter (ASCII code) is used as a subscript.

Example) When the axis name is X, a subscript is added as indicated below.

Setting value	Axis name displayed on a screen such as the position display screen
0	X
1	X1
77	XM
83	XS

If a multi-path system is used, no extended axis name is used within a path, and no subscript is set for the axis names, then the path number is automatically used as the subscript for the axis names. To disable

the display of axis name subscripts, set a blank (32) of ASCII code in the parameter for specifying an axis name subscript.

> **NOTE**
> If an extended axis name is used even for one axis within a path, the use of an axis name subscript becomes impossible within the path.

3141	Path name (1st character)
3142	Path name (2nd character)
3143	Path name (3rd character)
3144	Path name (4th character)
3145	Path name (5th character)
3146	Path name (6th character)
3147	Path name (7th character)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] See the character-code correspondence table.

Specify a path name with codes.
Any character string consisting of alphanumeric characters, katakana characters, and special characters with a maximum length of seven characters can be displayed as a series name.

> **NOTE**
> 1 For characters and codes, see the correspondence table in Appendix A.
> 2 When 0 is set in parameter No. 3141, PATH1(,PATH2...) are displayed as path names.

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE						

[Input type] Parameter input
[Data type] Bit path

 # 6 NPE With an M02, M30, or M99 block, program registration is assumed to be:
0: Completed
1: Not completed

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9				NE8

[Input type] Parameter input
[Data type] Bit path

 # 0 NE8 Editing of subprograms with program numbers 8000 to 8999
0: Not inhibited
1: Inhibited
When this parameter is set to 1, the following editing operations are disabled:
(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 8000 to 8999 are not deleted.)
(2) Program output (Even when outputting all programs is specified, programs with program numbers 8000 to 8999 are not output.)
(3) Program number search
(4) Program editing of registered programs
(5) Program registration
(6) Program collation
(7) Displaying programs

 # 4 NE9 Editing of subprograms with program numbers 9000 to 9999
0: Not inhibited
1: Inhibited
When this parameter is set to 1, the following editing operations are disabled:
(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)
(2) Program output (Even when outputting all programs is specified, programs with program numbers 9000 to 9999 are not output.)
(3) Program number search
(4) Program editing of registered programs
(5) Program registration
(6) Program collation
(7) Displaying programs

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL	MER	MZE					

[Input type] Parameter input
[Data type] Bit path

 # 5 MZE After MDI operation is started, program editing during operation is:
0: Enabled
1: Disabled

 # 6 MER When the last block of a program has been executed at single block operation in the MDI mode, the executed block is:
0: Not deleted
1: Deleted

> **NOTE**
> When MER is set to 0, the program is deleted if the end-of-record mark (%) is read and executed. (The mark % is automatically inserted at the end of a program.)

 # 7 MCL Whether a program prepared in the MDI mode is cleared by reset
0: Not deleted
1: Deleted

	#7	#6	#5	#4	#3	#2	#1	#0
3204		MKP						

[Input type] Parameter input
[Data type] Bit path

 # 6 MKP When M02, M03, or EOR (%) is executed in MDI operation, the created MDI program is automatically:
0: Deleted.
1: Not deleted.

> **NOTE**
> If bit 6 (MER) of parameter No. 3203 is set to 1, choose whether to automatically delete a created program when the last block is executed.

3210	Program protection (PSW)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 99999999

This parameter sets a password for protecting program Nos. 9000 to 9999. When a value other than zero is set in this parameter and this value differs from the keyword set in parameter No.3211, bit 4 (NE9) of parameter No.3202 for protecting program Nos. 9000 to 9999 is automatically set to 1.
This disables the editing of program Nos. 9000 to 9999. Until the value set as the password is set as a keyword, NE9 cannot be set to 0 and the password cannot be modified.

> **NOTE**
> 1 The state where password ≠ 0 and password ≠ keyword is referred to as the locked state. When an attempt is made to modify the password by MDI input operation in this state, the warning message "WRITE PROTECTED" is displayed to indicate that the password cannot be modified. When an attempt is made to modify the password with G10 (programmable parameter input), alarm (PS0231) is issued.
> 2 When the value of the password is not 0, the parameter screen does not display the password. Care must be taken in setting a password.

3211	Program protection key (KEY)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 99999999

When the value set as the password (set in parameter No.3210) is set in this parameter, the locked state is released and the user can now modify the password and the value set in bit 4 (NE9) of parameter No.3202.

> **NOTE**
> The value set in this parameter is not displayed. When the power is turned off, this parameter is set to 0.

3220	Password (PSW)

[Input type] Locked parameter
[Data type] 2-word
[Valid data range] 0 to 99999999

This parameter sets a password (PSW). When a value other than 0 is set, a password is set. When a password is set, a blank is displayed in this parameter, and the state (locked state) where an operation such as program editing is locked is set. When password (PSW) = 0, namely, in the normal state, or when password (PSW) = keyword (KEY), namely, in the unlock state, this parameter can be set.

3221	Keyword (KEY)

[Input type] Locked parameter
[Data type] 2-word
[Valid data range] 0 to 99999999

When the same value as the password (PSW) is set in this parameter, the lock is released (unlock state). The value set in this parameter is not displayed.
The value of this parameter is initialized to 0 automatically when the power is turned on. So, if the power is turned off in the unlock state then is turned on again, the lock state is automatically set.

3222	Program protection range (minimum value) (PMIN)

3223	Program protection range (maximum value) (PMAX)

[Input type] Locked parameter
[Data type] 2-word
[Valid data range] 0 to 99999999

The programs in a range set here can be locked. Set the minimum program number and maximum program number of a desired range.
Set these parameters to satisfy PMAX > PMIN.
These parameters can be set when password (PSW) = 0, namely, in the normal state, or when password (PSW) = keyword (KEY), namely in the unlock state.
Example)
Parameter No.3222 = 7000
Parameter No.3223 = 8499
When the values above are set, the programs from O7000 to O8499 can be locked.
When PMIN = 0, the specification of PMIN = 9000 is assumed. When PMAX = 0, the specification of PMAX = 9999 is assumed. So, when these parameters are set to the defaults, the programs from O9000 to O9999 are locked.

NOTE

1. Parameter No. 3220 to No. 3223 are neither punched nor read.
2. Parameter No. 3220 to No. 3223 are not cleared even when a parameter file clear operation is performed in the IPL state.
3. The values of a password (PSW) and keyword (KEY) are not displayed. When password (PSW) = 0, 0 is displayed in parameter No. 3220 to indicate that the normal state is set.
4. When a password (PSW) or keyword (KEY) is set, [+INPUT] has the same effect as [INPUT]. For example, if the input operation "1[+INPUT]" is performed when 99 is set in the keyword (KEY) parameter, "1" is set.

	#7	#6	#5	#4	#3	#2	#1	#0
3280								NLC

[Input type] Parameter input
[Data type] Bit

 # 0 NLC Dynamic display language switching is:
0: Enabled.
1: Disabled.
When dynamic display language switching is disabled, the language setting screen is not displayed. In this case, change the setting of parameter No. 3281 on the parameter screen then turn on the power again to switch the display language.

3281	Display language

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 14

Select a display language from the following:
0 : English
1 : Japanese
2 : German
3 : French
4 : Chinese
5 : Italian
6 : Korean
7 : Spanish
8 : Dutch
9 : Danish
10 : Portuguese
11 : Polish
12 : Hungarian
13 : Swedish
14 : Czech
If a number not indicated above is set, English is selected.

	#7	#6	#5	#4	#3	#2	#1	#0
3400			PGD				MGC	

[Input type] Parameter input
[Data type] Bit path

 # 1 MGC When a single block specifies multiple M commands, an M code group check is:
0: Made.
1: Not made.

 # 5 PGD The G10.9 command (programmable diameter/radius specification switching) is:

0: Disabled.
1: Enabled.

> **NOTE**
> 1 The option for the dynamic diameter/radius switching function is required.
> 2 When the G10.9 command is enabled by this parameter, signal-based dynamic diameter/radius switching is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB	ABS	MAB				DPI
			ABS	MAB				DPI

[Input type] Parameter input
[Data type] Bit path

 # 0 DPI When a decimal point is omitted in an address that can include a decimal point

0: The least input increment is assumed. (Normal decimal point input)
1: The unit of mm, inches, degree, or second is assumed. (Pocket calculator type decimal point input)

 # 4 MAB Switching between the absolute and incremental command modes in MDI operation is performed according to:

0: G90/G91.
1: Bit 5 (ABS) of parameter No. 3401.

> **NOTE**
> When G code system A is used with the lathe system, this parameter is invalid.

 # 5 ABS A programmed command in MDI operation is regarded as:

0: Incremental command.
1: Absolute command.

> **NOTE**
> Bit 5 (ABS) of parameter No. 3401 is valid when bit 4 (MAB) of parameter No. 3401 is set to 1.
> When G code system A is used with the lathe system, this parameter is invalid.

 # 6 GSB The G code system is set.
 # 7 GSC

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

NOTE
G code system B and G code system C are optional functions. When no option is selected, G code system A is used, regardless of the setting of these parameters.

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM	G91			G01
	G23	CLR			G91	G19	G18	G01

[Input type] Parameter input
[Data type] Bit path

 # 0 G01 G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

 # 1 G18 Plane selected when power is turned on or when the control is cleared
0: G17 mode (plane XY)
1: G18 mode (plane ZX)

 # 2 G19 Plane selected when power is turned on or when the control is cleared
0: The setting of bit 1 (G18) of parameter No. 3402 is followed.
1: G19 mode (plane YZ)
When this bit is set to 1, set bit 1 (G18) of parameter No. 3402 to 0.

 # 3 G91 When the power is turned on or when the control is cleared
0: G90 mode (absolute command)
1: G91 mode (incremental command)

 # 4 FPM At power-on time or in the cleared state:
0: G99 or G95 mode (feed per revolution) is set.
1: G98 or G94 mode (feed per minute) is set.

 # 6 CLR Reset button on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the User's Manual.

 # 7 G23 When the power is turned on
0: G22 mode (stored stroke check on)
1: G23 mode (stored stroke check off)

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B		M02	M30		SBP		

[Input type] Parameter input
[Data type] Bit path

 # 2 SBP In an external device subprogram call, the address P format is based on:
0: File number specification
1: Program number specification

NOTE
In memory card operation, the program number specification format is used, regardless of the setting of this parameter.

 # 4 M30 When M30 is specified in a memory operation:
0: M30 is sent to the machine, and the head of the program is automatically searched for. So, when the ready signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.
1: M30 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

 # 5 M02 When M02 is specified in memory operation
0: M02 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.
1: M02 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

 # 7 M3B The number of M codes that can be specified in one block
0: One
1: Up to three

	#7	#6	#5	#4	#3	#2	#1	#0
3405				CCR	G36		DWL	AUX
							DWL	AUX

[Input type] Parameter input
[Data type] Bit path

 # 0 AUX When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the multiplication factor for a value output (onto the code signal) relative to a specified value is such that:

0: The same multiplication factor is used for both of metric input and inch input.
1: A multiplication factor used for inch input is 10 times greater than that used for metric input.

When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the value output onto the code signal is a specified value multiplied by a value indicated below.

Increment system		Parameter AUX=0	Parameter AUX=1
Metric input system	IS-A for reference axis	100 times	100 times
	IS-B for reference axis	1000 times	1000 times
	IS-C for reference axis	10000 times	10000 times
	IS-D for reference axis	100000 times	100000 times
	IS-E for reference axis	1000000 times	1000000 times
Inch input system	IS-A for reference axis	100 times	1000 times
	IS-B for reference axis	1000 times	10000 times
	IS-C for reference axis	10000 times	100000 times
	IS-D for reference axis	100000 times	1000000 times
	IS-E for reference axis	1000000 times	10000000 times

 # 1 DWL The dwell time (G04) is:

0: Always dwell per second.
1: Dwell per second in the feed per minute mode (G94), or dwell per rotation in the feed per rotation mode (G95).

 # 3 G36 As a G code to be used with the automatic tool length measurement function (M series)/automatic tool offset function (T series) is:

0: G36 (T series only)/G37 is used.
1: G37.1/G37.2/G37.3 is used.

> **NOTE**
> If it is necessary to perform circular threading (counterclockwise), set this parameter to 1.

 # 4 CCR Addresses used for chamfering
0: Address is “I”, “J”, or “K”.
In direct drawing dimension programming, addresses ",C", ",R", and ",A" (with comma) are used in stead of "C", "R", and "A".
1: Address is “C”.
Addresses used for direct drawing dimension programming are "C", "R", and "A" without comma.

> **NOTE**
> If this bit (CCR) is set to 0, the function for changing the compensation direction by specifying I, J, or K in a G01 block in the cutter compensation/ tool nose radius compensation mode cannot be used.
> If this bit (CCR) is set to 1 when address C is used as an axis name, the chamfer function cannot be used.

	#7	#6	#5	#4	#3	#2	#1	#0
3406	C07	C06	C05	C04	C03	C02	C01	

	#7	#6	#5	#4	#3	#2	#1	#0
3407	C15	C14	C13	C12	C11	C10	C09	C08

	#7	#6	#5	#4	#3	#2	#1	#0
3408	C23	C22	C21	C20	C19	C18	C17	C16

	#7	#6	#5	#4	#3	#2	#1	#0
3409		C30	C29	C28	C27	C26	C25	C24

[Input type] Parameter input
[Data type] Bit

C01 to C30 If bit 6 (CLR) of parameter No. 3402 is set to 1, set a group of G codes to be placed in the cleared state when the CNC is reset by the reset key of the MDI panel, the external reset signal, the reset & rewind signal, or the emergency stop signal.
The table below indicates the correspondence between bits and G code groups
The setting of a bit has the following meaning:
0: Places the G code group in the cleared state.
1: Does not place G code group in the cleared state.

Parameter	G code group
C01	01
C02	02
C03	03
:	:
D30	30

3410	Tolerance of arc radius

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 999999999

When a circular interpolation command is executed, the tolerance for the radius between the start point and the end point is set.

3411	M code preventing buffering 1
3412	M code preventing buffering 2
	:
3420	M code preventing buffering 10

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 999999999

Set M codes that prevent buffering the following blocks. If processing directed by an M code must be performed by the machine without buffering the following block, specify the M code.
M00, M01, M02, and M30 always prevent buffering even when they are not specified in these parameters.

3421	Range specification 1 of M codes that do not perform buffering (lower limit)
3422	Range specification 1 of M codes that do not perform buffering (upper limit)
3423	Range specification 2 of M codes that do not perform buffering (lower limit)
3424	Range specification 2 of M codes that do not perform buffering (upper limit)
3425	Range specification 3 of M codes that do not perform buffering (lower limit)
3426	Range specification 3 of M codes that do not perform buffering (upper limit)
3427	Range specification 4 of M codes that do not perform buffering (lower limit)
3428	Range specification 4 of M codes that do not perform buffering (upper limit)
3429	Range specification 5 of M codes that do not perform buffering (lower limit)
3430	Range specification 5 of M codes that do not perform buffering (upper limit)
3431	Range specification 6 of M codes that do not perform buffering (lower limit)
3432	Range specification 6 of M codes that do not perform buffering (upper limit)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

When a specified M code is within the range specified with parameter Nos. 3421 and 3422, 3423 and 3424, 3425 and 3426, 3427 and 3428, 3429 and 3430, or 3431 and 3432, buffering for the next block is not performed until the execution of the block is completed.

> **NOTE**
> M00, M01, M02, and M30 are M codes that do not perform buffering, regardless of parameter setting. M98, M99, M codes for calling subprograms, and M codes for calling custom macros are M codes that performs buffering, regardless of parameter setting.

3441	Start number of M codes for which an M code group can be set (1)

3442	Start number of M codes for which an M code group can be set (2)

3443	Start number of M codes for which an M code group can be set (3)

3444	Start number of M codes for which an M code group can be set (4)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0, 100to99999999

Code numbers 0 to 99 on the M code group setting screen correspond to M00 to M99. When adding M codes after the first 100 M codes, specify a start M code number in these parameters. Thus, up to 400 M codes can be added to the M code group setting screen in groups of 100 M codes starting with the set value. When 0 is set, no M codes are added to the M code group setting screen.

When setting these parameters, follow the setting condition described below. If the condition is not satisfied, no M codes are added to the M code group setting screen as in the case where 0 is set.

(Setting condition)

The settings of parameters (1) to (4) (excluding the setting of 0) must satisfy:

99 < (1), (1)+99 < (2), (2)+99 < (39, (3) +99 < (4)

	#7	#6	#5	#4	#3	#2	#1	#0
3450	BDX							AUP

[Input type] Parameter input
[Data type] Bit path

 # 0 AUP The second auxiliary function specified in the calculator-type decimal point input format, with a decimal point, or with a negative value is:
0: Disabled.
1: Enabled.
If the second auxiliary function is specified after setting this bit to 0, the following operation results:
1. When a value is specified without a decimal point
   A specified value is output onto the code signal without modification, regardless of the setting of the calculator-type decimal point input format (with bit 0 (DPI) of parameter No. 3401).
2. When a value is specified with a decimal point
   The alarm (PS0007) is issued.
3. When a negative value is specified
   The alarm (PS0006) is issued.

 # 7 BDX When ASCII code is called using the same address as the address for the second auxiliary function (specified by parameter No. 3460), this parameter prevents the argument unit used when the option for the second auxiliary function is selected from differing from the argument unit used when the same option is not selected.
0: When bit 0 (AUP) of parameter No. 3450 is set to 1, the argument unit differs, depending on whether the option for the second auxiliary function is selected or not.
1: The same argument unit is used. (The unit applied when the option for the second auxiliary function is selected is used.)

[Example]
A setting is made so that address B is used to call O9004, and the program O1 below is executed with parameter No. 3460 = 66.

```
O1      O9004
B2      #500 = #146
M30         M99
```

When the increment system is IS-B, and metric input is used, #500 assumes a value indicated in the table below.

Bit 0 (DPI) of parameter No. 3401	Bit 0 (AUP) of parameter No. 3450	BDX=0		BDX=1
		Without the second auxiliary function option	With the second auxiliary function option	
0	0	2.000	2.000	2.000
	1	2.000	0.002	0.002
1	0	2.000	2.000	2.000
	1	2.000	2.000	2.000

	#7	#6	#5	#4	#3	#2	#1	#0
3451								
								GQS

[Input type] Parameter input
[Data type] Bit path

 # 0 GQS When threading is specified, the threading start angle shift function (Q) is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3452	EAP							

[Input type] Parameter input
[Data type] Bit path

 # 7 EAP When bit 0 (ADX) of parameter No. 3455 is set to 1, calculator-type decimal point input at a macro calling argument address is:
0: Enabled.
1: Disabled.

> **NOTE**
> This parameter is valid when bit 0 (DPI) of parameter No. 3401 is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3455								AXDx

[Input type] Parameter input
[Data type] Bit axis

 # 0 AXDx If a decimal point is omitted for an axis address with which a decimal point can be used, the value is determined:
0: In accordance with the least input increment. (Normal decimal point input)
1: In millimeters, inches, or seconds. (calculator-type decimal point input)

> **NOTE**
> This parameter specifies the calculator-type decimal point input function for each axis.
> For the same axis name, be sure to make the same setting.

	#7	#6	#5	#4	#3	#2	#1	#0
3457	SCF				SYS	MC1	MC2	LIB

[Input type] Parameter input
[Data type] Bit path

NOTE
1 The parameters LIB, MC2, MC1, and SYS are used to set a search folder for the following subprogram/macro calls:
- Subprogram call based on an M code
- Subprogram call based on a particular address
- Subprogram call based on a second auxiliary function code
- Macro call based on a G code
- Macro call based on an M code
- Macro call based on a T code
- One-touch macro call

2 The parameter SCF is used to set whether to add a search folder for the following subprogram/macro calls:
- Subprogram call based on M98
- Figure copy based on G72.1/G72.2
- Macro call based on G65/G66/G66.1
- Macro interrupt based on M96

 # 0 LIB The common program directory "//CNC_MEM/USER/LIBRARY/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

 # 1 MC2 MTB dedicated directory 2 "//CNC_MEM/MTB2/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

 # 2 MC1 MTB dedicated directory 1 "//CNC_MEM/MTB1/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

 # 3 SYS The system directory "//CNC_MEM/SYSTEM/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

 # 7 SCF A search folder is:
0: Not added.
1: Added.

When a search folder is added, a search is made in the following order:
1) Folder where the main program is stored
2) Common program folder, which is an initial folder
3) MTB-dedicated folder 2, which is an initial folder
4) MTB-dedicated folder 1, which is an initial folder
5) System folder, which is an initial folder

The folders of 3) through 5) can be excluded from search target folders by setting the parameters MC2, MC1, and SYS.

3460	Second auxiliary function specification address

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 65to67, 85to87

Specify which of A, B, C, U, V, and W is to be used as the address for specifying the second auxiliary function. If an address used as an axis name is specified, the second auxiliary function is disabled.

Name	A	B	C	U	V	W
Setting value	65	66	67	85	86	87

Address B is assumed when a value other than the above is set.
However, the name U, V, or W can be used with the T series only when G code system B or C is used. When a value from 85 to 87 is specified with G code system A, the specification address for the second auxiliary function is B.

3471	Allowable difference between the specified end position and the end position obtained from the increase/decrease and frequency in spiral interpolation or conic interpolation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 to 999999999

This parameter sets the maximum allowable difference (absolute value) between the specified end position and the end position obtained from the increase/decrease and frequency in spiral or conic interpolation.

3472	Minimum radius needed to maintain the actual speed in spiral or conic interpolation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] (For IS-B and millimeter machines, 1.0 to 999999.999; for inch machines, 1.0 to 99999.9999)

If this parameter value is 0 or a value outside the valid data range, the minimum value of the range is assumed.

In spiral interpolation and conic interpolation, the speed is generally held constant. In an area near the center, the spiral radius decreases, resulting in an extremely high angular velocity. To prevent this, once the spiral radius has reached the parameter-set value, the angular velocity subsequently remains constant. As a result, the actual speed decreases.

	#7	#6	#5	#4	#3	#2	#1	#0
3605								BDPx

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 BDPx Both-direction pitch error compensation is:
0: Not used.
1: Used.

3620	Number of the pitch error compensation position for the reference position for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023

Set the number of the pitch error compensation position for the reference position for each axis.

3621	Number of the pitch error compensation position at extremely negative position for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023

Set the number of the pitch error compensation position at the extremely negative position for each axis.

3622	Number of the pitch error compensation position at extremely positive position for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023

Set the number of the pitch error compensation position at the extremely positive position for each axis.
This value must be larger than set value of parameter (No.3620).

3623	Magnification for pitch error compensation for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 100

Set the magnification for pitch error compensation for each axis.
If the magnification is set to 1, the same unit as the detection unit is used for the compensation data.

3624	Interval between pitch error compensation positions for each axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

The pitch error compensation positions are arranged with equal spacing. The space between two adjacent positions is set for each axis.
The minimum interval between pitch error compensation positions is limited and obtained from the following equation:
Minimum interval between pitch error compensation positions = maximum feedrate/7500
Unit : mm, inch, deg or mm/min, inch/min, deg/min
Example:

When the maximum feedrate is 15000 mm/min, the minimum interval between pitch error compensation positions is 2 mm.

3625	Travel distance per revolution in pitch error compensation of rotation axis type

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

If the pitch error compensation of rotation axis type is performed (bit 1 (ROSx) of parameter No. 1006 is set to 0 and bit 0 (ROTx) of parameter No. 1006 is set to 1), set the travel distance per revolution. The travel distance per revolution does not have to be 360 degrees, and a cycle of pitch error compensation of rotation axis type can be set.
However, the travel distance per revolution, compensation interval, and number of compensation points must satisfy the following condition:
(Travel distance per revolution) = (Compensation interval) × (Number of compensation points)
The compensation at each compensation point must be set so that the total compensation per revolution equals 0.

> **NOTE**
> If 0 is set, the travel distance per revolution becomes 360 degrees.

3626	Number of the both-direction pitch error compensation position at extremely negative position (for movement in the negative direction)

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023, 3000 to 4023

When using both-direction pitch error compensation, set the number of compensation point at the farthest end in the negative direction for a movement in the negative direction.

> **NOTE**
> 1 For a movement in the positive direction, set the compensation point number at the farthest end in the negative direction in parameter No. 3621.
> 2 A set of compensation data items for a single axis should not be set to lie astride 1023 to 3000.

3627	Pitch error compensation at reference position when a movement to the reference position is made from the direction opposite to the direction of reference position return

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -32768 to 32767

Set the absolute value of pitch error compensation at reference position when a movement to the reference position is made from the negative direction if the direction of reference position return (bit 5 (ZMI) of parameter No. 1006) is positive or from the positive direction if the direction of reference position return is negative.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	CRF

[Input type] Parameter input
[Data type] Bit path

 # 0 CRF Reference position setting at an arbitrary position under Cs contour control is:
0: Not used.
1: Used.

> **NOTE**
> When this function is used, an attempt to specify G00 for a Cs contour control axis without performing a reference position return operation even once after switching the serial spindle to the Cs contour control mode results in the alarm (PS0303) even if bit 1 (NRF) of parameter No. 3700 is set to 0. Be sure to perform a reference position return operation by specifying G28.

 # 1 NRF With the first move command (G00) after switching the series spindle to Cs contour control mode:
0: A reference position return operation is once performed then positioning is performed.
1: A normal positioning operation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Input type] Parameter input
[Data type] Bit path

 # 1 EMS The multi-spindle control function is:
0: Used.
1: Not used.

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 A/Ss Spindle motor type is :
0: Analog spindle. (Prohibition of use)
1: Serial spindle.

> **NOTE**
> 1 When an analog spindle is used, the option for spindle analog output is required.
> 2 When a serial spindle is used, the option for spindle serial output is required.
> 3 The option for the number of controlled spindles needs to be specified.

3717	Motor number to each spindle

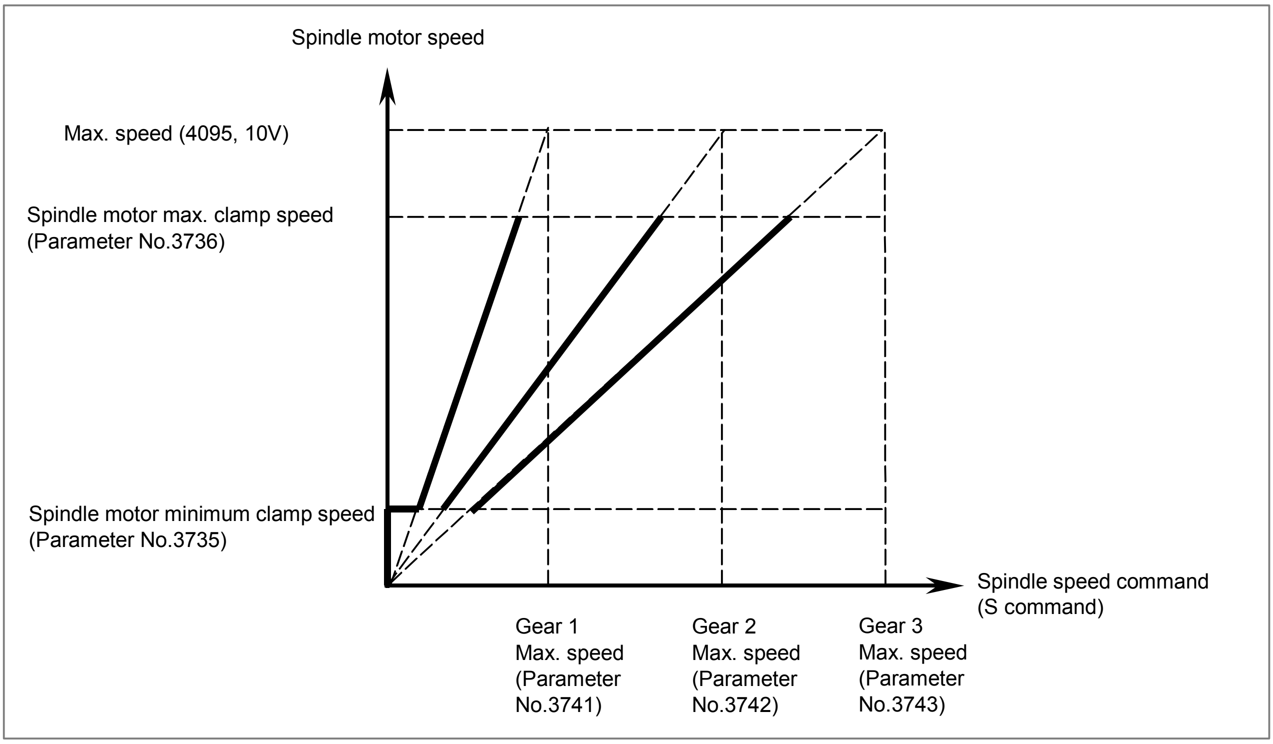
> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes
Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

Set the maximum spindle speed corresponding to each gear.



3770	Axis as the calculation reference in constant surface speed control

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the axis as the calculation reference in constant surface speed control.

> **NOTE**
> When 0 is set, constant surface speed control is always applied to the X-axis. In this case, specifying P in a G96 block has no effect on the constant surface speed control.

3781	P code for selecting the spindle in multi-spindle control

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767

If bit 3 (MPP) of parameter No. 3703 is set to 1, set the P code to select each spindle under multi-spindle control. Specify the P code in a block containing the S command.

Example)
If the P code value for selecting the second spindle is set to 3,
S1000 P3;
causes the second spindle to rotate at S1000.

> **NOTE**
> 1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
> 2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
> 3 Under multipath control, the P code specified here is valid for each path.
> For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
> 4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
> 5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
> 6 To use this parameter, the multi-spindle control function is needed.

Parameters Nos. 4000 to 4799 are basically used with the serial spindle amplifier (SPM). For details of these parameters, refer to either of the following manuals and other related documents, depending on the spindle that is actually connected.

- FANUC AC SPINDLE MOTOR αi series Parameter Manual (B-65280EN)

	#7	#6	#5	#4	#3	#2	#1	#0
4900								FLRs

[Input type] Parameter input
[Data type] Bit spindle

 # 0 FLRs When the spindle speed fluctuation detection function is used, the unit of an allowable ratio (q) and fluctuation ratio (r) set by parameter No. 4911 and No. 4912 is:
0: 1%
1: 0.1%

4911	Allowable speed ratio (q) used to assume that the spindle has reached a specified speed

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, set an allowable speed ratio (q) used to assume that the spindle has reached a specified speed.

> **NOTE**
> The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

4912	Spindle variation ratio (r) for not issuing a spindle speed fluctuation detection alarm

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000

When the spindle speed fluctuation detection function is used, set a spindle fluctuation ratio (r) for not issuing an alarm.

> **NOTE**
> The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

4913	Spindle speed fluctuation width (i) for not issuing a spindle speed fluctuation detection alarm

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999

When the spindle speed fluctuation detection function is used, set an allowable fluctuation width (i) for not issuing an alarm.

4914	Time (p) from the change of a specified speed until spindle speed fluctuation detection is started

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] msec
[Valid data range] 0 to 99999

When the spindle speed fluctuation detection function is used, set a time (p) from the change of a specified speed until spindle speed fluctuation detection is started. In other words, spindle speed fluctuation detection is not performed until a set time has elapsed after a specified speed is changed. However, when the actual spindle speed is assumed to have reached a specified value within a set time (p), spindle speed fluctuation detection is started.

	#7	#6	#5	#4	#3	#2	#1	#0
4950						ISZs	IDMs	IORs

[Input type] Parameter input
[Data type] Bit spindle

 # 0 IORs Resetting the system in the spindle positioning mode
0: Does not releases the mode.
1: Releases the mode

 # 1 IDMs The direction of spindle positioning (half-fixed angle positioning based on M codes) is:
0: Plus direction.
1: Minus direction.

 # 2 ISZs When an M code for switching to the spindle positioning mode is specified for spindle positioning:
0: The spindle is switched to the spindle positioning mode, and spindle orientation operation is performed.
1: Only the switching of the spindle to the spindle positioning mode is performed. (Spindle orientation operation is not performed.)

4960	M code specifying the spindle orientation

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 97

Set an M code for switching to the spindle positioning mode.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4961	M code releasing the spindle positioning mode

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 97

Set an M code for canceling the spindle positioning mode on the spindle positioning axis.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4962	M code for specifying a spindle positioning angle

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 9999999

Two methods are available for specifying spindle positioning. One method uses axis address for arbitrary-angle positioning. The other use an M code for half-fixed angle positioning. This parameter sets an M code for the latter method.

In this parameter, set an M code to be used for half-fixed angle positioning based on M codes.

Six M code from Mα to M(α+5) are used for half-fixed angle positioning, when a is the value of this parameter.

- When the number of M codes is set in parameter No. 4964, let α be the value set in parameter No. 4962, and let β be the value set in parameter No. 4964. Then, β M codes from Mα to M(α+β-1) are used as M codes for half-fixed angle positioning based on M codes.

The table below indicates the relationship between the M codes and positioning angles.

M code	Positioning angle	Example: Positioning angle when θ = 30°
Mα	θ	30°
M(α+1)	2θ	60°
M(α+2)	3θ	90°
M(α+3)	4θ	120°
M(α+4)	5θ	150°
M(α+5)	6θ	180°
:	:	:
M(α+β-1)	$\beta\times\theta$	$\beta\times$30°

β represents the number of M codes set in parameter No. 4964.
(When parameter No. 4964 is set to 0, β = 6.)
θ represents the basic angular displacement set in parameter No.4963.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4963	Basic angle for half-fixed angle positioning

[Input type] Parameter input
[Data type] Real spindle
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 to 60

This parameter sets a basic angular displacement used for half-fixed angle positioning using M codes.

4964	Number of M codes for specifying a spindle positioning angle

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 255

This parameter sets the number of M codes used for Half-fixed angle positioning using M codes.

As many M codes as the number specified in this parameter, starting with the M code specified in parameter No.4962, are used to specify half-fixed angle positioning.

Let α be the value of parameter No.4962, and let β be the value of parameter No.4964. That is, M codes from Ma to M(α+β-1) are used for half-fixed angle positioning.

Setting this parameter to 0 has the same effect as setting 6. That is, M code from Mα to M(α+5) are used for half-fixed angle positioning.

NOTE

1 Make sure that M codes from Ma to M (α+β-1) do not duplicate other M codes.
2 Do not set an M code that duplicates other M codes used for spindle positioning.
3 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO						
		EVO			TAL		TLB	TLC

[Input type] Parameter input
[Data type] Bit path

 # 0 TLC
 # 1 TLB These bits are used to select a tool length compensation type.

Type	TLB	TLC
Tool length compensation A	0	0
Tool length compensation B	1	0
Tool length compensation C	-	1

The axis to which cutter compensation is applied varies from type to type as described below.
Tool length compensation A :
Z-axis at all times
Tool length compensation B :
Axis perpendicular to a specified plane (G17/G18/G19)
Tool length compensation C :
Axis specified in a block that specifies G43/G44

 # 3 TAL Tool length compensation C
0: Generates an alarm when two or more axes are offset
1: Not generate an alarm even if two or more axes are offset

 # 6 EVO If a tool compensation value modification is made for tool length compensation A or tool length compensation B in the offset mode (G43 or G44):
0: The new value becomes valid in a block where G43, G44, or an H code is specified next.
1: The new value becomes valid in a block where buffering is performed next.

	#7	#6	#5	#4	#3	#2	#1	#0
5002						LWT	LGN	

[Input type] Parameter input
[Data type] Bit path

 # 1 LGN Geometry offset number of tool offset
0: Is the same as wear offset number
1: Specifies the geometry offset number by the tool selection number

NOTE
This parameter is valid when the option for tool geometry compensation or tool wear compensation is specified.

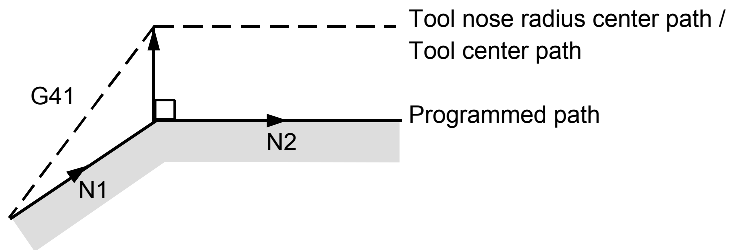
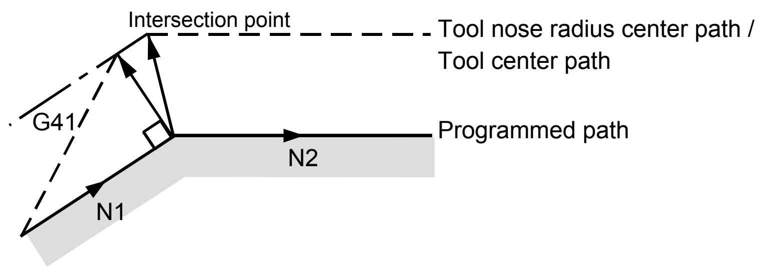
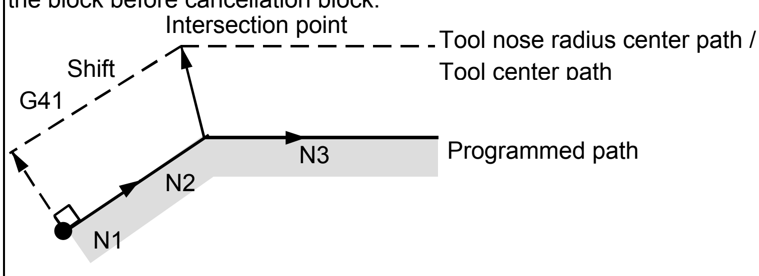
 # 2 LWT Tool wear compensation is performed by:
0: Moving the tool.
1: Shifting the coordinate system.

NOTE
This parameter is valid when the option for tool geometry compensation or tool wear compensation is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
5003							SUV	SUP
							SUV	SUP

[Input type] Parameter input
[Data type] Bit path

 # 0 SUP
 # 1 SUV These bits are used to specify the type of startup/cancellation of cutter compensation or tool nose radius compensation.

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output. 
0	1	Type B	A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output. 
1	0 1	Type C	When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.  When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.

NOTE
When SUV,SUP = 0,1 (type B), an operation equivalent to that of FS16i-T is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	
						ODI		

[Input type] Parameter input
[Data type] Bit path

 # 1 ORC The setting of a tool offset value is corrected as:
0: Diameter value
1: Radius value

> **NOTE**
> This parameter is valid only for an axis based on diameter specification. For an axis based on radius specification, specify a radius value, regardless of the setting of this parameter.

 # 2 ODI The setting of a cutter compensation/tool-nose radius compensation value is corrected as:
0: Radius value
1: Diameter value

	#7	#6	#5	#4	#3	#2	#1	#0
5008				MCR				

[Input type] Parameter input
[Data type] Bit path

 # 4 MCR If G41/G42 (cutter compensation or tool nose radius compensation) is specified in the MDI mode, an alarm is:
0: Not raised.
1: Raised. (alarm PS5257)

5028	Number of digits of an offset number used with a T code command

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Specify the number of digits of a T code portion that is used for a tool offset number (wear offset number when the tool geometry/wear compensation function is used).
When 0 is set, the number of digits is determined by the number of tool compensation values.
When the number of tool compensation values is 1 to 9: Lower 1 digit
When the number of tool compensation values is 10 to 99: Lower 2 digits
When the number of tool compensation values is 100 to 999: Lower 3 digits

Example :

When an offset number is specified using the lower 2 digits of a T code, set 2 in parameter No. 5028.

Txxxxxx yy

xxxxxx : Tool selection

yy : Tool offset number

> **NOTE**
> A value longer than the setting of parameter No. 3032 (allowable number of digits of a T code) cannot be set.

5029	Number of tool compensation value memories common to paths

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Word

[Valid data range] 0 to 999

When using memories common to paths, set the number of common tool compensation values in this parameter.

Ensure that the setting of this parameter does not exceed the number of tool compensation values set for each path (parameter No. 5024).

[Example 1]

When parameter No. 5029 = 10, parameter No. 5024 (path 1) = 15, and parameter No. 5024 (path 2) = 30 in a 2-path system, tool compensation numbers 1 to 10 of all paths are made common.

[Example 2]

When parameter No. 5029 = 20 and the other conditions are the same as for Example 1, tool compensation numbers 1 to 15 are made common.

> **NOTE**
> 1 When a multi-path system involving the machining center system and lathe system is used, memories are made common in each system.
> 2 In each of the machining center system and lathe system, the same unit of tool compensation values needs to be used.
> 3 Ensure that the setting of parameter No. 5029 does not exceed the number of tool compensation values for each path (parameter No. 5024). If the setting of parameter No. 5029 exceeds the number of compensation values of a path, the least of the numbers of compensation values in all paths is made common.
> 4 When 0 or a negative value is set, memories common to paths are not used.

	#7	#6	#5	#4	#3	#2	#1	#0
5040					TCT			OWD

[Input type] Parameter input
[Data type] Bit path

 # 0 OWD In radius programming (bit 1 (ORC) of parameter No. 5004 is set to 1),
0: Tool offset values of both geometry compensation and wear compensation are specified by radius.
1: Tool offset value of geometry compensation is specified by radius and tool offset value of wear compensation is specified by diameter, for an axis of diameter programming.

NOTE
This parameter is valid when the option for tool geometry compensation or tool wear compensation is specified.

 # 3 TCT The tool change method is based on:
0: Turret rotation. (Tool change operation is performed with a T command only.)
With a T command, an auxiliary function and tool offset operation are performed.
1: Automatic tool changer (ATC).
(Tool change operation is performed with an M command (such as M06)).
With a T command, an auxiliary function only is performed.
This parameter is valid with the T series only.

5042	#7	#6	#5	#4	#3	#2	#1	#0
					OFE	OFD	OFC	OFA

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 OFA
 # 1 OFC
 # 2 OFD
 # 3 OFE These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFE	OFD	OFC	OFA	Unit	Valid data range
0	0	0	1	0.01mm	±9999.99mm
0	0	0	0	0.001mm	±9999.999mm
0	0	1	0	0.0001mm	±9999.9999mm
0	1	0	0	0.00001mm	±9999.99999mm
1	0	0	0	0.000001mm	±999.999999mm

For inch input

OFE	OFD	OFC	OFA	Unit	Valid data range
0	0	0	1	0.001inch	±999.999inch
0	0	0	0	0.0001inch	±999.9999inch
0	0	1	0	0.00001inch	±999.99999inch
0	1	0	0	0.000001inch	±999.999999inch
1	0	0	0	0.0000001inch	±99.9999999inch

5101	#7	#6	#5	#4	#3	#2	#1	#0
								FXY
								FXY

[Input type] Parameter input
[Data type] Bit path

 # 0 FXY The drilling axis in the drilling canned cycle is:
0: Always the Z-axis
1: The axis selected by the program

> **NOTE**
> In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 15 format.

	#7	#6	#5	#4	#3	#2	#1	#0
5200						CRG		G84
						CRG		G84

[Input type] Parameter input
[Data type] Bit path

 # 0 G84 Method for specifying rigid tapping
0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No.5210).
1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

 # 2 CRG Rigid mode when a rigid mode cancel command is specified (G80, G01 group G code, reset, etc.)
0: Canceled after rigid tapping signal RGTAP is set to "0".
1: Canceled before rigid tapping signal RGTAP is set to "0".

	#7	#6	#5	#4	#3	#2	#1	#0
5202					CHR			

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

 # 3 CHR When the option for interpolation type rigid tapping is available:
0: Interpolation type rigid tapping is selected.
1: Conventional rigid tapping is selected.
This parameter is valid when the option for interpolation type rigid tapping is available. When the option for interpolation type rigid tapping is not available, conventional rigid tapping is selected, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5203							HRM	HRG

[Input type] Parameter input
[Data type] Bit path

 # 0 HRG Rigid tapping by the manual handle is:
0: Disabled.
1: Enabled.

 # 1 HRM When the tapping axis moves in the negative direction during rigid tapping controlled by the manual handle, the direction in which the spindle rotates is determined as follows:

0: In G84 mode, the spindle rotates in a normal direction. In G74 mode, the spindle rotates in reverse.

1: In G84 mode, the spindle rotates in reverse. In G74 mode, the spindle rotates in a normal direction.

5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)
5244	Maximum spindle speed in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 9999

Spindle position coder gear ratio
1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999

Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.

Set the same value for both parameter No.5241 and parameter No.5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200 will be issued. This applies to the M series.

	#7	#6	#5	#4	#3	#2	#1	#0
5400	SCR	XSC				D3R		

[Input type] Parameter input
[Data type] Bit path

 # 2 D3R The three-dimensional coordinate conversion mode can be cancelled by:

0: The G69 (M series) command, the G69.1 (T series) command, a reset operation, or a CNC reset by signal input from the PMC.

1: The G69 (M series) command or G69.1 (T series) command only.

 # 6 XSC The setting of a scaling magnification (axis-by-axis scaling) is:

0: Disabled.

1: Enabled.

 # 7 SCR Scaling (G51) magnification unit
0: 0.00001 times (1/100,000)
1: 0.001 times

	#7	#6	#5	#4	#3	#2	#1	#0
5401								**SCLx**

[Input type] Parameter input
[Data type] Bit axis

 # 0 SCLx Scaling on this axis
0: Invalidated
1: Validated

5411	**Scaling (G51) magnification**

[Input type] Setting input
[Data type] 2-word path
[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No.5400)
[Valid data range] 1to999999999
This parameter sets a scaling magnification when axis-by-axis scaling is disabled (with bit 6 (XSC) of parameter No. 5400 set to 0). If no scaling magnification (P) is specified in the program, the setting of this parameter is used as a scaling magnification.

> **NOTE**
> When bit 7 (SCR) of parameter No. 5400 is set to 1, the valid data range is 1 to 9999999.

5412	**Rapid traverse rate for a hole machining cycle in three-dimensional coordinate conversion mode**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
This parameter sets a rapid traverse rate for a hole machining cycle in the three-dimensional coordinate conversion mode.

5421	Scaling magnification for each axis

[Input type] Setting input
[Data type] 2-word axis
[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No.5400)
[Valid data range] -999999999 to –1, 1 to 999999999

This parameter sets a scaling magnification for each axis when axis-by-axis scaling is enabled (with bit 6 (XSC) of parameter No. 5400 set to 1). For the first spindle to the third spindle (X-axis to Z-axis), the setting of this parameter is used as a scaling magnification if scaling magnifications (I, J, K) are not specified in the program.

> **NOTE**
> When bit 7 (SCR) of parameter No. 5400 is set to 1, the valid data ranges are -9999999 to -1 and 1 to 9999999.

	#7	#6	#5	#4	#3	#2	#1	#0
5431								MDL

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 MDL The G60 code (one-direction positioning) is:
0: One-shot G code (group 00).
1: Modal G code (group 01).

5440	Positioning direction and overrun distance in single directional positioning

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] -32767 to 32767

This parameter sets the positioning direction and overrun distance in single directional positioning (G60) for each axis. The positioning direction is specified using a setting data sign, and the overrun distance using a value set here.
Overrun distance>0: The positioning direction is positive (+).
Overrun distance<0: The positioning direction is negative (*).
Overrun distance=0: Single directional positioning is not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5450						PLS		

[Input type] Parameter input
[Data type] Bit path

 # 2 PLS The polar coordinate interpolation shift function is:
0: Not used.
1: Used.
This enables machining using the workpiece coordinate system with a desired point which is not the center of the rotation axis set as the origin of the coordinate system in polar coordinate interpolation.

5460	Axis (linear axis) specification for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of linear axis to execute polar interpolation.

5461	Axis (rotation axis) specification for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of rotation axis to execute polar interpolation.

5463	Automatic override tolerance ratio for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100
Typical setting: 90% (treated as 90% when set to 0)
Set the tolerance ratio of the fastest cutting feedrate to the speed of the rotation axis during automatic override of polar coordinate interpolation.

5464	Compensation for error on hypothetical axis of polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(For IS-B, -999999.999 to +999999.999)

This parameter is used to set the error if the center of the rotation axis on which polar coordinate interpolation is performed is not on the X-axis.

If the setting of the parameter is "0", regular polar coordinate interpolation is performed.

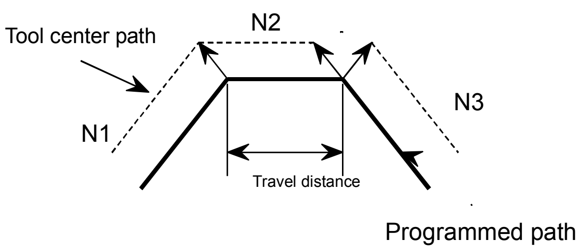
5481	Feedrate of rotation of the normal direction controlled axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/min
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)

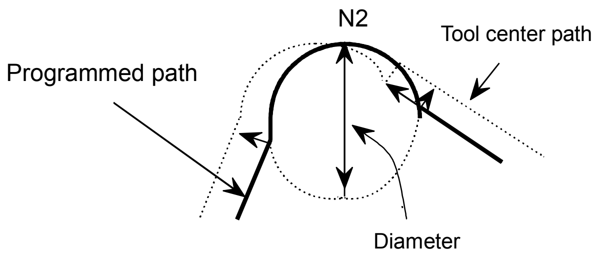
This parameter sets the feedrate of the movement along the normal direction controlled axis that is inserted at the start point of a block during normal direction control.

5483	Limit value of movement that is executed at the normal direction angle of a preceding block

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to standard parameter setting table (B)



For straight line
When the travel distance of N2 in the figure on the left does not exceed the setting, block N2 is machined with the tool being normal to block N1.



For arc
When the arc diameter of N2 in the figure on the left does not exceed the setting, arc N2 is machined with the tool being normal to block N1. A normal direction axis is not controlled to move in the normal direction according to the arc movement.

	#7	#6	#5	#4	#3	#2	#1	#0
5630								SPN

[Input type] Parameter input
[Data type] Bit path

 # 0 SPN The amount of linear axis division (span value) in exponential interpolation is:
0: Specified with parameter No.5643.
1: Specified using address K in a block containing G02.3/G03.3. When address K is not specified, the value set with parameter No.5643 is used.

5641	Linear axis number subject to exponential interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets the ordinal number, among the controlled axes, for the linear axis to which exponential interpolation is applied.

5642	Rotation axis number subject exponential interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes

This parameter sets the ordinal number, among the controlled axes, for the rotation axis to which exponential interpolation is applied.

5643	Amount of linear axis division (span value) in exponential interpolation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 999999999

This parameter sets an amount of linear axis division in exponential interpolation when bit 0 (SPN) of parameter No. 5630 is set to 0 or when address K is not specified.

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM	HGO			MGO	G67
			SBM	HGO	V15		MGO	G67

[Input type] Parameter input
[Data type] Bit path

 # 0 G67 If the macro continuous-state call cancel command (G67) is specified when the macro continuous-state call mode (G66/G66.1) is not set:
0: Alarm PS0122 is issued.
1: The specification of G67 is ignored.

 # 1 MGO When a GOTO statement for specifying custom macro control is executed, a high-speed branch to 20 sequence numbers executed from the start of the program is:
0: A high-speed branch is not caused to n sequence numbers from the start of the executed program.
1: A high-speed branch is caused to n sequence numbers from the start of the program.

 # 3 V15 As system variable numbers for tool offset:
0 : The standard system variable numbers for the Series 16 are used.
1 : The same system variable numbers as those used for the Series 15 are used.

The tables below indicate the system variables for tool offset numbers 1 to 999. The values for tool offset numbers 1 to 200 can be read from or assigned to the system variables in parentheses.

(1) Tool offset memory A

	System variable number	
	V15 = 0	V15 = 1
Wear offset value	#10001 to #10999 (#2001 to #2200)	#10001 to #10999 (#2001 to #2200)

(2) Tool offset memory B

	System variable number	
	V15 = 0	V15 = 1
Geometry offset value	#11001 to #11999 (#2201 to #2400)	#10001 to #10999 (#2001 to #2200)
Wear offset value	#10001 to #10999 (#2001 to #2200)	#11001 to #11999 (#2201 to #2400)

(3) Tool offset memory C

		System variable number	
		V15 = 0	V15 = 1
Tool length offset	Geometry offset value	#11001 to #11999 (#2201 to #2400)	#10001 to #10999 (#2001 to #2200)
	Wear offset value	#10001 to #10999 (#2001 to #2200)	#11001 to #11999 (#2201 to #2400)
Tool radius offset	Geometry offset value	#13001 to #13999	#12001 to #12999
	Wear offset value	#12001 to #12999	#13001 to #13999

 # 4 **HGO** When a GOTO statement in a custom macro control command is executed, a high-speed branch to the 30 sequence numbers immediately before the executed statement is:

0: Not made.

1: Made.

 # 5 **SBM** Custom macro statement

0: Not stop the single block

1: Stops the single block

If you want to disable the single blocks in custom macro statements using system variable #3003, set this parameter to 0. If this parameter is set to 1, the single blocks in custom macro statements cannot be disabled using system variable #3003. To control single blocks in custom macro statements using system variable #3003, use bit 7 (SBV) of parameter No. 6000.

	#7	#6	#5	#4	#3	#2	#1	#0
6001		CCV	TCS	CRO	PV5		PRT	MIF

[Input type] Parameter input
[Data type] Bit path

 # 0 MIF The custom macro interface signals are based on:
0: Standard specification.
(The signals UI000 to UI015, UO000 to UO015, and UO100 to UO131 are used.)
1: Extended specification.
(The signals UI000 to UI031, UI100 to UI131, UI200 to UI231, UI300 to UI331, UO000 to UO031, UO100 to UO131, UO200 to UO231, and UO300 to UO331 are used.)

 # 1 PRT Reading zero when data is output using a DPRINT command
0: Outputs a space
1: Outputs no data

 # 3 PV5 Custom macro common variables:
0: #500 to #549 are output. (Note 1)
1: #100 to #149 and #500 to 549 are output. (Note 1)

> **NOTE**
> The variables depend on the selected options.

		Custom macro common variable addition option	
		Not selected	**Selected**
Embedded macro option	**Not selected**	#500 to #549 or #100 to #149 and #500 to #549	#500 to #999 or #100 to #199 and #500 to #999
	Selected	#500 to #549 or #100 to #499 and #500 to #549	#500 to #999 or #100 to #49 and #500 to #999

 # 4 CRO ISO code in BPRWT or DPRNT command
0: Outputs only “LF” after data is output
1: Outputs “LF” and “CR” after data is output

 # 5 TCS Custom macro (subprogram)
0: Not called using a T code
1: Called using a T code

 # 6 CCV Common variables #100 to #149(NOTE) cleared by power-off are:
0: Cleared to <null>
1: Not cleared

> **NOTE**
> The variables depend on the selected options.

		Custom macro common variable addition option	
		Not selected	Selected
Embedded macro option	Not selected	#100to#149	#100to#199
	Selected	#100to#499	

	#7	#6	#5	#4	#3	#2	#1	#0
6003			MSB	MPR	TSE	MIN		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 2 MIN Custom macro interrupt
0: Performed by interrupting an in-execution block (Custom macro interrupt type I)
1: Performed after an in-execution block is completed (Custom macro interrupt type II)

 # 3 TSE Custom macro interrupt signal UINT
0: Edge trigger method (Rising edge)
1: Status trigger method

 # 4 MPR Custom macro interrupt valid/invalid M code
0: M96/M97
1: M code set using parameters (Nos. 6033 and 6034)

 # 5 MSB Interrupt program
0: Uses a dedicated local variable (Macro-type interrupt)
1: Uses the same local variable as in the main program (Subprogram- type interrupt)

	#7	#6	#5	#4	#3	#2	#1	#0
6004						VHD		NAT
			D15					NAT

[Input type] Parameter input
[Data type] Bit path

 # 0 NAT The results of the custom macro functions ATAN (with 2 arguments) and ASIN are specified as follows:
0: The result of ATAN is 0 to 360.0.
The result of ASIN is 270.0 to 0 to 90.0.
1: The result of ATAN is -180.0 to 0 to 180.0.
The result of ASIN is -90.0 to 0 to 90.0.

 # 2 VHD With system variables #5121 to #5140:
0: The tool offset value (geometry offset value) in the block currently being executed is read. (This parameter is valid only when tool geometry/tool wear compensation memories are available.)
1: An interrupt travel distance based on manual handle interrupt is read.

 # 5 D15 When tool compensation memory C is used, for reading or writing tool offset values (for up to offset number 200) for D code (tool radius), the same system variables, #2401 through #2800, as Series 15 are:
0: Not used.
1: Used.
When bit 3 (V15) of parameter No. 6000 is set to 1

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2401	[#_OFSDG[1]]	#2601	[#_OFSDW[1]]
2	#2402	[#_OFSDG[2]]	#2602	[#_OFSDW[2]]
3	#2403	[#_OFSDG[3]]	#2603	[#_OFSDW[3]]
:	:	:	:	:
199	#2599	[#_OFSDG[199]]	#2799	[#_OFSDW[199]]
200	#2600	[#_OFSDG[200]]	#2800	[#_OFSDW[200]]

	#7	#6	#5	#4	#3	#2	#1	#0
6007				CVA	MGE	BCS	SCS	DPG

[Input type] Parameter input
[Data type] Bit path

 # 0 DPG Specifies whether to allow G codes with a decimal point to be called.
0: Do not allow.
1: Allow.

 # 1 SCS Specifies whether to call subprograms with S codes.
0: Do not call with S codes.
1: Call with S codes.

 # 2 BCS Specifies whether to call subprograms with the second auxiliary function codes.
0: Do not call with the second auxiliary function codes.
1: Call with the second auxiliary function codes.

 # 3 MGE Specifies whether a G code modal call is made after movement or for each block.
0: Make a call for each block (equivalent to G66.1).
1: Make a call after movement (equivalent to G66).

 # 4 CVA The format for macro call arguments is specified as follows:
0: Arguments are passed in NC format without modifications.
1: Arguments are converted to macro format then passed.
Example)
When G65 P_ X10 ; is specified, the value in local variable #24 in the calling program is set as follows:

Command	CVA=0	CVA=1
#24	0.01	0.01
ADP[#24]	10.0	0.01

NOTE
External operations are the same unless the ADP function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
6008	IJK	GMP		ISO			MCA	F16

[Input type] Parameter input
[Data type] Bit path

 # 0 F16 The precision of operation is based on:
0: New specification.
1: FS16i compatible specification.

 # 1 MCA A macro alarm specification based on system variable #3000 is selected as follows:
0: An alarm number obtained by adding 3000 to a value assigned to variable #3000 and the corresponding message are displayed. (A value from 0 to 200 can be assigned to variable #3000.)
1: A value assigned to variable #3000 and the corresponding message are displayed. (A value from 0 to 4095 can be assigned to variable #3000.)
(Example)
Execution of #3000=1 (ALARM MESSAGE);
When bit 1 (MCA) of parameter No. 6008 is set to 0:
The alarm screen displays "3001 ALARM MESSAGE".
When bit 1 (MCA) of parameter No. 6008 is set to 1:
The alarm screen displays "MC0001 ALARM MESSAGE".

 # 4 ISO
0: When the EIA code is used, the bit patters of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameter No. 6010 to No. 6018.
1: When the ISO/ASCII code is used, the bit patters of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameter No. 6010 to No. 6018.

 # 6 GMP The calling of M, S, T, a second auxiliary function code, or a particular code during the calling of a G code, and the calling of a G code during the calling of M, S, T, a second auxiliary function code, or particular code are:

0: Not allowed. (They are executed as an ordinary G, M, S, T, second auxiliary function code, and NC address.)

1: Allowed.

#7 IJK For addresses I, J, and K specified as arguments:

0: Argument specification I or II is automatically determined.

1: Argument specification I is always used.

> **Example**
> When K_J_I_ is specified:
> - When this parameter is set to 0:
>   Argument specification II is used and K=#6, J=#8, and I=#10 are specified.
> - When this parameter is set to1:
>   Argument specification I is used and I=#4, J=#5, and K=#6 are specified regardless of the spcification order.
>   (Argument specification II cannot be used.)

	#7	#6	#5	#4	#3	#2	#1	#0
6010	*7	*6	*5	*4	*3	*2	*1	*0

	#7	#6	#5	#4	#3	#2	#1	#0
6011	=7	=6	=5	=4	=3	=2	=1	=0

	#7	#6	#5	#4	#3	#2	#1	#0
6012	#7	#6	#5	#4	#3	#2	#1	#0

	#7	#6	#5	#4	#3	#2	#1	#0
6013	[7	[6	[5	[4	[3	[2	[1	[0

	#7	#6	#5	#4	#3	#2	#1	#0
6014	]7	]6	]5	]4	]3	]2	]1	]0

	#7	#6	#5	#4	#3	#2	#1	#0
6015	?7	?6	?5	?4	?3	?2	?1	?0

	#7	#6	#5	#4	#3	#2	#1	#0
6016	@7	@6	@5	@4	@3	@2	@1	@0

	#7	#6	#5	#4	#3	#2	#1	#0
6017	&7	&6	&5	&4	&3	&2	&1	&0

	#7	#6	#5	#4	#3	#2	#1	#0
6018	_7	_6	_5	_4	_3	_2	_1	_0

[Input type] Parameter input
[Data type] Bit path

*0 to *7 : The bit pattern of the EIA or ISO/ASCII code indicating * is set.
=0 to =7 : The bit pattern of the EIA or ISO/ASCII code indicating = is set.
#0 to #7 : The bit pattern of the EIA or ISO/ASCII code indicating # is set.
[0 to [7 : The bit pattern of the EIA or ISO/ASCII code indicating [is set.
]0 to]7 : The bit pattern of the EIA or ISO/ASCII code indicating] is set.
?0 to ?7 : The bit pattern of the EIA or ISO/ASCII code indicating ? is set.
@0 to @7 : The bit pattern of the EIA or ISO/ASCII code indicating @ is set.
&0 to &7 : The bit pattern of the EIA or ISO/ASCII code indicating & is set.
_0 to _7 : The bit pattern of the EIA or ISO/ASCII code indicating _ is set.
0: A corresponding bit is 0.
1: A corresponding bit is 1.

6030	M code to execute external device subprogram calls

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 99999999

Set the M code to execute external device subprogram calls. When 0 is set, M198 is used. M01, M02, M30, M98, and M99 cannot be used to execute external device subprogram calls. When a negative number, 1, 2, 30, 98, or 99 is set for this parameter, M198 is used to execute external device subprogram calls.

6031	Start number of common variables to be protected among the common variables (#500 to #999)

6032	End number of common variables to be protected among the common variables (#500 to #999)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 500 to 999

Among the common variables (#500 to #999), the range of common variables specified by this parameter can be protected (by setting their attributes to read-only). If a write attempt (on the left side) is made, an alarm is issued.

> **NOTE**
> Set 0 in both parameter No. 6031 and No. 6032 not to protect common variables.

6033	M code that validates a custom macro interrupt

6034	M code that invalidates a custom macro interrupt

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 03 to 99999999 (excluding 30, 98 and 99)

These parameters can be used when MPR, #4 of parameter No.6003, is 1. M96 is used as a valid M code and M97 is used as an invalid M code when MPR is 0, irrespective of the state of this parameter.

6036	Number of custom macro variables common to tool path (for #100 to #199 (#499))

[Input type] Parameter input
[Data type] Word system common
[Valid data range] 0 to 400

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #100 to #199 (up to #499 in a system with the embedded macro option) may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example
When 20 is set in parameter No. 6036
#100 to #119: Shared by all paths
#120 to #149: Used by each path independently

Example)
When 20 is set in parameter No. 6036
#100 to #119: Shared by all paths
#120 to #149: Used by each path independently

NOTE
1. To use up to #199, the option for adding custom macro common variables is required.
2. To use up to #499, the embedded macro option is required.
3. When 0 or a negative value is set, the memory common to paths is not used.

6037	Number of custom macro variables common to tool path (for #500 to #999)

[Input type] Parameter input
[Data type] Word system common
[Valid data range] 0 to 500

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #500 to #999 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example
When 50 is set in parameter No. 6037
#500 to #549: Shared by all paths #120 to #149:
#550 to #599: Used by each path independently

> **NOTE**
> 1 To use up to #999, the option for adding custom macro common variables is required.
> 2 When 0 or a negative value is set, the memory common to paths is not used.

6038	Start G code used to call a custom macro

[Input type] Parameter input
[Data type] Word path
[Valid data range] -9999 to 9999

6039	Start program number of a custom macro called by G code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6040	Number of G codes used to call custom macros

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255

Set this parameter to define multiple custom macro calls using G codes at a time. With G codes as many as the value set in parameter No. 6040 starting with the G code set in parameter No. 6038, the custom macros of program numbers as many as the value set in parameter No. 6040 starting with the program number set in parameter No. 6039 can be called. Set 0 in parameter No. 6040 to disable this mode of calling.

If a negative value is set in parameter No. 6038, the modal call mode is entered. Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

Example)

When parameter No. 6038 = 900, parameter No. 6039 = 1000, and parameter No. 6040 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:

G900 → O1000
G901 → O1001
G902 → O1002
:
G999 → O1099

When the setting of parameter No. 6038 is changed to -900, the same set of custom macro calls (modal calls) is defined.

NOTE

1 When the following conditions are satisfied, all calls using these parameters are disabled:
   1) When a value not within the specifiable range is set in each parameter
   2) (Value of parameter No.6039 + value of parameter No.6040 - 1) > 9999
2 The specification of a mixture of simple calls and modal calls is not allowed.
3 If a range of G codes set by these parameters duplicate G codes specified in parameter No.6050 to No.6059, the calls specified by parameter No.6050 to 6059 are made preferentially.

6041	Start G code with a decimal point used to call a custom macro

[Input type] Parameter input
[Data type] Word path
[Valid data range] -999 to 999

6042	Start program number of a custom macro called by G code with a decimal point

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6043	Number of G codes with a decimal point used to call custom macros

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255

Set this parameter to define multiple custom macro calls using G codes with a decimal point at a time. With G codes with a decimal point as many as the value set in parameter No. 6043 starting with the G code with a decimal point set in parameter No. 6041, the custom macros of program numbers as many as the value set in parameter No. 6043 starting with the program number set in parameter No. 6042 can be called. Set 0 in parameter No. 6043 to disable this mode of calling.
If a negative value is set in parameter No. 6041, the modal call mode is entered. Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

Example)

When parameter No. 6041 = 900, parameter No. 6042 = 2000, and parameter No. 6043 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:
G90.0 → O2000
G90.1 → O2001
G90.2 → O2002

:
G99.9 → O2099
When the setting of parameter No. 6041 is changed to -900, the same set of custom macro calls (modal calls) is defined.

NOTE
1 When the following conditions are satisfied, all calls using these parameters are disabled:
  1) When a value not within the specifiable range is set in each parameter
  2) (Value of parameter No.6042 + value of parameter No.6043 - 1) > 9999
  3) When bit 0 (DPG) of parameter No. 6007 = 0 (to disable calls using G codes with a decimal point)
2 The specification of a mixture of simple calls and modal calls is not allowed.
3 If a range of G codes set by these parameters duplicate G codes specified in parameter No.6060 to No.6069, the calls specified by parameter No.6060 to 6069 are made preferentially.

6044	Start M code used to call a subprogram

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

6045	Start program number of a subprogram called by M code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6046	Number of M codes used to call subprograms (number of subprograms called by M codes)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767
Set this parameter to define multiple subprogram calls using M codes at a time. With M codes as many as the value set in parameter No. 6046 starting with the M code set in parameter No. 6044, the subprograms of program numbers as many as the value set in parameter No. 6046 starting with the program number set in 6045 can be called. Set 0 in parameter No. 6046 to disable this mode of calling.

Example)
When parameter No. 6044 = 80000000, parameter No. 6045 = 3000, and parameter No. 6046 = 100 are set, a set of 100 subprogram calls is defined as follows:

M80000000 → O3000
M80000001 → O3001
M80000002 → O3002
:
M80000099 → O3099

NOTE
1 When the following conditions are satisfied, all calls using these parameters are disabled:
1) When a value not within the specifiable range is set in each parameter
2) (Value of parameter No. 6045 + value of parameter No. 6046 - 1) > 9999
2 If a range of M codes set by these parameters duplicate M codes specified in parameter No. 6071 to No. 6079, the calls specified by parameter No. 6071 to 6079 are made preferentially.

6047	Start M code used to call a custom macro

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

6048	Start program number of a custom macro called by M code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6049	Number of M codes used to call custom macros (number of custom macros called by M codes)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767

Set this parameter to define multiple custom macro calls using M codes at a time. With M codes as many as the value set in parameter No. 6049 starting with the M code set in parameter No. 6047, the custom macros of program numbers as many as the value set in parameter No. 6049 starting with the program number set in parameter No. 6048 can be called. Set 0 in parameter No. 6049 to disable this mode of calling.

Example)

When parameter No. 6047 = 90000000, parameter No. 6048 = 4000, and parameter No. 6049 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:

M90000000 → O4000
M90000001 → O4001
M90000002 → O4002

:
M90000099 → O4099

> **NOTE**
> 1 When the following conditions are satisfied, all calls using these parameters are disabled:
>   1) When a value not within the specifiable range is set in each parameter
>   2) (Value of parameter No. 6048 + value of parameter No. 6049 - 1) > 9999
> 2 If a range of M codes set by these parameters duplicate M codes specified in parameter No. 6080 through No. 6089, the calls specified by parameter No. 6080 through 6089 are made preferentially.
> 3 When a 5-digit or longer O number is used, the option for 8-digit program numbers is required.

6050	**G code that calls the custom macro of program number 9010**
to	
6059	**G code that calls the custom macro of program number 9019**

[Input type] Parameter input
[Data type] Word path
[Valid data range] (-9999 to 9999 : excluding 0, 5, 65, 66 and 67)

Set the G codes used to call the custom macros of program numbers 9010 to 9019. However, note that when a negative value is set in this parameter, it becomes a modal call. For example, if this parameter is set to -11, the modal call mode is entered by G11.
Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

6060	**G code with a decimal point used to call the custom macro of program number 9040**
to	
6069	**G code with a decimal point used to call the custom macro of program number 9049**

[Input type] Parameter input
[Data type] Word path
[Valid data range] -999 to 999

Set the G codes used to call the custom macros of program numbers 9040 to 9049. However, note that when a negative value is set in this parameter, it becomes a modal call. For example, if this parameter is set to -11, the modal call mode is entered by G1.1.

Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007. Set G codes in the format Gm.n. The value expressed by $(m\times10+n)$ is set in the parameter. The values m and n must satisfy the following relationships: $0 \le m \le 99$, $0 \le n \le 9$.

6071	**M code used to call the subprogram of program number 9001**
to	
6079	**M code used to call the subprogram of program number 9009**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)

These parameters set the M codes that call the subprograms of program numbers 9001 to 9009.

NOTE

If the same M code is set in these parameters, the younger number is called preferentially. For example, if 100 is set in parameter No. 6071 and 6072, and programs O9001 and O9002 both exist, O9001 is called when M100 is specified.

6080	**M code used to call the custom macro of program number 9020**
to	
6089	**M code used to call the custom macro of program number 9029**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)

Set the M codes used to call the custom macros of program numbers 9020 to 9029. The simple call mode is set.

> **NOTE**
> 1 If the same M code is set in these parameters, the younger number is called preferentially. For example, if 200 is set in parameter No. 6081 and No. 6082, and programs O9021 and O9022 both exist, O9021 is called when M200 is specified.
> 2 If the same M code is set in a parameter (No. 6071 to No. 6079) used to call subprograms and in a parameter (No. 6080 to No. 6089) used to call custom macros, a custom macro is called preferentially. For example, if 300 is set in parameter No. 6071 and No. 6081, and programs O9001 and O9021 both exist, O9021 is called when M300 is specified.

6090	ASCII code that calls the subprogram of program number 9004

6091	ASCII code that calls the subprogram of program number 9005

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 65(A:41H) to 90(Z:5AH)

These parameters set the ASCII codes that call subprograms in decimal.
The settable addresses are indicated below.

Address	Parameter setting value	T series	M series
A	65	O	O
B	66	O	O
D	68	X	O
F	70	O	O
H	72	O	O
I	73	O	O
J	74	O	O
K	75	O	O
L	76	O	O
M	77	O	O
P	80	O	O
Q	81	O	O
R	82	O	O
S	83	O	O
T	84	O	O
V	86	X	O
X	88	X	O
Y	89	X	O
Z	90	X	O

> **NOTE**
> 1 When address L is set, the number of repeats cannot be specified.
> 2 Set 0 when no subprogram is called.

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF			HSS				

[Input type] Parameter input
[Data type] Bit path

 # 4 HSS

0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

 # 7 SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1

[Input type] Parameter input
[Data type] Bit path

1S1 to 1S8 These parameters specify whether to enable or disable each high-speed skip signal when the G31 skip command is issued.
The following table shows the correspondence between the bits, input signals, and commands.
The settings of the bits have the following meaning :
0: The high-speed skip signal corresponding to a bit is disabled.
1: The high-speed skip signal corresponding to a bit is enabled.

Parameter	High-speed skip signals
1S1	HDI0
1S2	HDI1
1S3	HDI2
1S4	HDI3
1S5	HDI4
1S6	HDI5
1S7	HDI6
1S8	HDI7

NOTE
Do not specify the same signal simultaneously for different paths.

	#7	#6	#5	#4	#3	#2	#1	#0
6203	2S8	2S7	2S6	2S5	2S4	2S3	2S2	2S1

	#7	#6	#5	#4	#3	#2	#1	#0
6204	3S8	3S7	3S6	3S5	3S4	3S3	3S2	3S1

	#7	#6	#5	#4	#3	#2	#1	#0
6205	4S8	4S7	4S6	4S5	4S4	4S3	4S2	4S1

	#7	#6	#5	#4	#3	#2	#1	#0
6206	DS8	DS7	DS6	DS5	DS4	DS3	DS2	DS1

[Input type] Parameter input
[Data type] Bit path

1S1to1S8, 2S1to2S8, 3S1to3S8, 4S1to4S8, DS1toDS8

Specify which skip signal is enabled when the skip command (G31, or G31P1 to G31P4) and the dwell command (G04, G04Q1 to G04Q4) are issued with the multi-step skip function.
The following table shows the correspondence between the bits, input signals, and commands.
The setting of the bits have the following meaning :
0: The skip signal corresponding to a bit is invalid.
1: The skip signal corresponding to a bit is enabled.

Multi-step skip function					
Command / Input signal	G31 G31P1 G04Q1	G31P2 G04Q2	G31P3 G04Q3	G31P4 G04Q4	G04
SKIP/HDI0	1S1	2S1	3S1	4S1	DS1
SKIP2/HDI1	1S2	2S2	3S2	4S2	DS2
SKIP3/HDI2	1S3	2S3	3S3	4S3	DS3
SKIP4/HDI3	1S4	2S4	3S4	4S4	DS4
SKIP5/HDI4	1S5	2S5	3S5	4S5	DS5
SKIP6/HDI5	1S6	2S6	3S6	4S6	DS6
SKIP7/HDI6	1S7	2S7	3S7	4S7	DS7
SKIP8/HDI7	1S8	2S8	3S8	4S8	DS8

NOTE
HDI0 to HDI7 are high-speed skip signals.

	#7	#6	#5	#4	#3	#2	#1	#0
6208	9S8	9S7	9S6	9S5	9S4	9S3	9S2	9S1

[Input type] Parameter input
[Data type] Bit path

9S1 to 9S8 Specify which high-speed skip signal is enabled for the continuous high-speed skip command G31P90 or the EGB skip command G31.8.
The settings of each bit have the following meaning:
0: The high-speed skip signal corresponding to the bit is disabled.
1: The high-speed skip signal corresponding to the bit is enabled.
The bits correspond to signals as follows:

Parameter	High-speed skip signal
9S1	HDI0
9S2	HDI1
9S3	HDI2
9S4	HDI3
9S5	HDI4
9S6	HDI5
9S7	HDI6
9S8	HDI7

6254	ε value on the X axis during automatic tool compensation (T series)
	ε value during automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)

6255	ε value on the Z axis during automatic tool compensation (T series)
	ε value during automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the relevant ε value during automatic tool compensation (T series) or automatic tool length measurement (M series).

NOTE
1 For the M series, when the setting of parameter No. 6255 or 6256 is 0, the setting of parameter No. 6254 is used.
2 Set a radius value regardless of whether diameter or radius programming is specified.

6287	Positional deviation limit in torque limit skip

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 327670

This parameter sets a positional deviation limit for each axis imposed when torque limit skip is specified. When the actual positional deviation exceeds the positional deviation limit, the alarm (SV0004) is issued and an immediate stop takes place.

6581	RGB value of color palette 1 for text
to	
6595	RGB value of color palette 15 for text

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 151515

Each of these parameters sets the RGB value of each color palette for text by specifying a 6-digit number as described below.
rrggbb: 6-digit number (rr: red data, gg: green data, bb: blue data)
The valid data range of each color is 0 to 15 (same as the tone levels on the color setting screen). When a number equal to or greater than 16 is specified, the specification of 15 is assumed.
Example)
When the tone level of a color is: red:1 green:2, blue:3, set 10203 in the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
6700								PCM

[Input type] Parameter input
[Data type] Bit path

 # 0 PCM M code that counts the total number of machined parts and the number of machined parts
0: M02, or M30, or an M code specified by parameter No.6710
1: Only M code specified by parameter No.6710

6710	M code that counts the number of machined parts

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 999999999

The total number of machined parts and the number of machined parts are counted (+1) when the M code set is executed.

> **NOTE**
> The setting of 0 is invalid (no count operation is performed with M00.) Moreover, M98, M99, M198 (external device subprogram calling), and M codes used for subprogram calling and macro calling cannot be set as M codes for count-up operation. (Even when such an M code is set, count-up operation is not performed, ignoring the M code.)

6711	Number of machined parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

The number of machined parts is counted (+1) together with the total number of machined parts when the M02, M30, or a M code specified by parameter No.6710 is executed.

NOTE
The number of parts is not counted for M02, M03, when bit 0 (PCM) of parameter No. 6700 is set to 1.

6712	Total number of machined parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

This parameter sets the total number of machined parts.
The total number of machined parts is counted (+1) when M02, M30, or an M code specified by parameter No.6710 is executed.

NOTE
The number of parts is not counted for M02, M30, when bit 0 (PCM) of parameter No. 6700 is set to 1.

6713	Number of required parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999

This parameter sets the number of required machined parts.
Required parts finish signal PRTSF <F0062#7> is output to PMC when the number of machined parts reaches the number of required parts. The number of parts is regarded as infinity when the number of required parts is zero. The PRTSF signal is then not output.

6750	Integrated value of power-on period

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of power-on period.

6751	Operation time (integrated value of time during automatic operation) 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999

For details, see the description of parameter No. 6752.

6752	Operation time (integrated value of time during automatic operation) 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of time during automatic operation (neither stop nor hold time included).
The actual time accumulated during operation is the sum of this parameter No. 6751 and parameter No. 6752.

6753	Integrated value of cutting time 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999

For details, see the description of parameter No. 6754.

6754	Integrated value of cutting time 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999

This parameter displays the integrated value of a cutting time that is performed in cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03).
The actual time accumulated during cutting is the sum of this parameter No. 6753 and parameter No. 6754.

	#7	#6	#5	#4	#3	#2	#1	#0
6801						LVF		

[Input type] Parameter input
[Data type] Bit path

 # 2 LVF When the life of a tool is counted in terms of time with the tool management function, the tool life count override signals *TLV0 to *TLV9<G049#0 to G050#1> are:
0: Invalid.
1: Valid.

6930	Maximum value of the operating range of the 1-st position switch (PSW101)
6931	Maximum value of the operating range of the 2-nd position switch (PSW102)
	:
6945	Maximum value of the operating range of the 16-th position switch (PSW116)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the maximum value of the operating range of the first to sixteenth position switches.

NOTE
1. For a diameter-specified axis, use radius values to specify the parameters used to set the maximum and minimum values of an operating range.
2. The position switch function is enabled upon completion of reference position return.

6950	Minimum value of the operating range of the 1-st position switch (PSW201)
6951	Minimum value of the operating range of the 2-nd position switch (PSW202)
	:
6965	Minimum value of the operating range of the 16-th position switch (PSW216)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the minimum value of the operating range of the first to sixteenth position switches.

NOTE
1. For a diameter-specified axis, use radius values to specify the parameters used to set the maximum and minimum values of an operating range.
2. The position switch function is enabled upon completion of reference position return.

	#7	#6	#5	#4	#3	#2	#1	#0
7001						JST		

[Input type] Parameter input
[Data type] Bit path

 # 2 JST In manual numerical specification, the STL signal indicating that automatic operation is being started is:
0: Not output.
1: Output.

	#7	#6	#5	#4	#3	#2	#1	#0
7002					JBF	JTF	JSF	JMF

[Input type] Parameter input
[Data type] Bit path

 # 0 JMF In manual numerical specification, M function specification is:
0: Allowed.
1: Not allowed.

 # 1 JSF In manual numerical specification, S function specification is:
0: Allowed.
1: Not allowed.

 # 2 JTF In manual numerical specification, T function specification is:
0: Allowed.
1: Not allowed.

 # 3 JBF In manual numerical specification, B function specification is:
0: Allowed.
1: Not allowed.

	#7	#6	#5	#4	#3	#2	#1	#0
7055					BCG			

[Input type] Parameter input
[Data type] Bit path

 # 3 BCG The bell-shaped acceleration/deceleration time constant change function is:
0: Disabled.
1: Enabled.

7066	Acceleration/deceleration reference speed for the bell-shaped acceleration/deceleration time constant change function

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
This parameter is used when the pre-interpolation bell-shaped acceleration/deceleration time constant change function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
7100							THD	JHD

[Input type] Parameter input
[Data type] Bit path

 # 0 JHD Manual handle feed in JOG feed mode or incremental feed in the manual handle feed
0: Invalid
1: Valid

 # 1 THD In the TEACH IN JOG mode, the manual pulse generator is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

[Input type] Parameter input
[Data type] Bit axis

 # 0 HNGx Axis movement direction for rotation direction of manual pulse generator
0: Same in direction
1: Reverse in direction

	#7	#6	#5	#4	#3	#2	#1	#0
7103					HIT	HNT	RTH	

[Input type] Parameter input
[Data type] Bit path

 # 1 RTH By a reset or emergency stop, the amount of manual handle interruption is:
0: Not canceled.
1: Canceled.

 # 2 **HNT** When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals) (MP1, MP2), the travel distance magnification for incremental feed/manual handle feed is:

0: Same.
1: 10 times greater.

 # 3 **HIT** When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals (MP1, MP2), the travel distance magnification for manual handle interrupt is:

0: Same.
1: 10 times greater.

7117	Allowable number of pulses that can be accumulated during manual handle feed

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Pulse
[Valid data range] 0 to 999999999

This parameter sets the number of pulses from the manual pulse generator that exceed the rapid traverse rate and can be accumulated without being discarded if manual handle feed faster than the rapid traverse rate is specified.

0:
The feedrate is clamped to the rapid traverse rate. Those handle pulses that exceed the rapid traverse rate are ignored. (The scale reading of the manual pulse generator may not match the travel distance.)

Other than 0:
The feedrate is clamped to the rapid traverse rate. However, those handle pulses that exceed the rapid traverse rate are not ignored. In connection with the manual handle feed travel distance selection signals MP1 and MP2 <G019#4, #5>, the incremental feed amount is determined as described below. (Even if the rotation of the manual pulse generator is stopped, the tool stops after moving by the number of pulses accumulated in the CNC.)

Let m be the magnification based on MP1 and MP2 <G019#4, #5>, and let n be the value set in parameter No. 7117. Then, the manual handle increment feed amount is:

When $n < m$:
Clamped to the value set in parameter No. 7117.

When $n \geq m$:
Clamped to a multiple of the selected magnification.

	#7	#6	#5	#4	#3	#2	#1	#0
7200		OP7	OP6	OP5	OP4	OP3	OP2	OP1

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 OP1 Mode selection on software operator's panel
0: Not performed
1: Performed

 # 1 OP2 JOG feed axis select and manual rapid traverse select on software operator's panel
0: Not performed
1: Performed

 # 2 OP3 Manual pulse generator's axis select and manual pulse generator's magnification select on software operator's panel
0: Not performed
1: Performed

 # 3 OP4 JOG feedrate override select, feedrate override select, and rapid traverse override select on software operator's panel
0: Not performed
1: Performed

 # 4 OP5 Optional block skip select, single block select, machine lock select, and dry run select on software operator's panel
0: Not performed
1: Performed

 # 5 OP6 Protect key on software operator's panel
0: Not performed
1: Performed

 # 6 OP7 Feed hold on software operator's panel
0: Not performed
1: Performed

7210 **Job-movement axis and its direction on software operator's panel "↑"**

7211 **Job-movement axis and its direction on software operator's panel "↓"**

7212 **Job-movement axis and its direction on software operator's panel "→"**

7213 **Job-movement axis and its direction on software operator's panel "←"**

7214 **Job-movement axis and its direction on software operator's panel "⇙"**

7215 **Job-movement axis and its direction on software operator's panel "⇗"**

7216 **Job-movement axis and its direction on software operator's panel "⤴"**

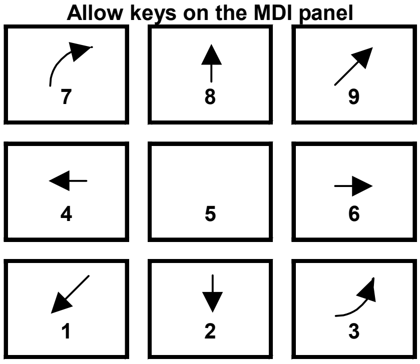
7217 **Job-movement axis and its direction on software operator's panel "↷"**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 8

On software operator's panel, set a feed axis corresponding to an arrow key on the MDI panel when jog feed is performed.

Setting value	Feed axis and direction
0	Not moved
1	First axis, positive direction
2	First axis, negative direction
3	Second axis, positive direction
4	Second axis, negative direction
5	Third axis, positive direction
6	Third axis, negative direction
7	Fourth axis, positive direction
8	Fourth axis, negative direction

Allow keys on the MDI panel



Example)

Under X, Y, and Z axis configuration, to set arrow keys to feed the axes in the direction specified as follows, set the parameters to the values given below. [8↑] to the positive direction of the Z axis, [2↓] to the negative direction of the Z axis, [6→] to the positive direction of the X axis [4←] to the negative direction of the X axis, [1⇙] to the positive direction of the Y axis, [9⇗] to the negative direction of the Y axis

Parameter No.7210 = 5 (Z axis, positive direction)
Parameter No.7211 = 6 (Z axis, negative direction)
Parameter No.7212 = 1 (X axis, positive direction)
Parameter No.7213 = 2 (X axis, negative direction)
Parameter No.7214 = 3 (Y axis, positive direction)
Parameter No.7215 = 4 (Y axis, negative direction)
Parameter No.7216 = 0 (Not used)
Parameter No.7217 = 0 (Not used)

	#7	#6	#5	#4	#3	#2	#1	#0
7300	MOU	MOA						

[Input type] Parameter input
[Data type] Bit path

 # 6 MOA In program restart operation, before movement to a machining restart point:
0: The last M, S, T, and B codes are output.
1: All M codes and the last S, T, and B codes are output.
This parameter is enabled when the MOU parameter is set to 1.

 # 7 MOU In program restart operation, before movement to a machining restart point after restart block search:
0: The M, S, T, and B codes are not output.
1: The last M, S, T, and B codes are output.

7310	Ordinal number of an axis along which a movement is made in dry run after program restart

[Input type] Setting input
[Data type] Byte axis
[Valid data range] 1 to (Number of controlled axes)
This parameter sets the ordinal number of an axis along which a movement is made in dry run after the program is restarted.

	#7	#6	#5	#4	#3	#2	#1	#0
7600	PLZ							

[Input type] Parameter input
[Data type] Bit path

 # 7 PLZ Reference position return based on a G28 command on the tool rotation axis for polygon turning is:
0: Performed in the same sequence as manual reference position return.
1: Performed by positioning using the rapid traverse rate.
The synchronous axis returns to the reference position in the same sequence as the manual reference position return when no return-to-reference position is performed after the power is turned on.

7610	Control axis number of tool rotation axis for polygon turning

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes

This parameter sets the control axis number of a rotation tool axis used for polygon turning.
However, when a G51.2 command is executed by setting 0 in this parameter, operation stops with the alarm (PS0314).

7640	Master axis in spindle-spindle polygon turning

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Within a path)

This parameter sets the master axis in spindle-spindle polygon turning.

NOTE
1. Spindle-spindle polygon turning is enabled only for serial spindles.
2. When any one of parameter No. 7640 and No. 7641 is set to 0, polygon turning is performed using the first spindle (master axis) and the second spindle (polygon synchronous axis) in the path to which the parameter belongs.
3. When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control option is required to specify an S command for the master axis.
4. When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.

7641	Polygon synchronous axis in spindle-spindle polygon turning

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Within a path)

This parameter sets the polygon synchronous (slave) axis in spindle-spindle polygon turning.

NOTE

1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 When any one of parameter No. 7640 and No. 7641 is set to 0, polygon turning is performed using the first spindle (master axis) and the second spindle (polygon synchronous axis) in the path to which the parameter belongs.
3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control option is required to specify an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.

7642	Master axis in spindle-spindle polygon turning (spindle number common to the system)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Common to the system)

This parameter sets the master axis in spindle-spindle polygon turning.

NOTE
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 This parameter is invalid if either parameter No. 7642 or No.7643 is set to 0. In this case, the settings of parameter No. 7640 and No.7641 are valid.
3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control option is required to specify an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
5 A spindle number common to the system is to be set in this parameter. When using this parameter, set 0 in parameter No. 7640 and No. 7641.

7643	Polygon synchronous axis in spindle-spindle polygon turning

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Common to the system)

This parameter sets the polygon synchronous (slave) axis in spindle-spindle polygon turning.

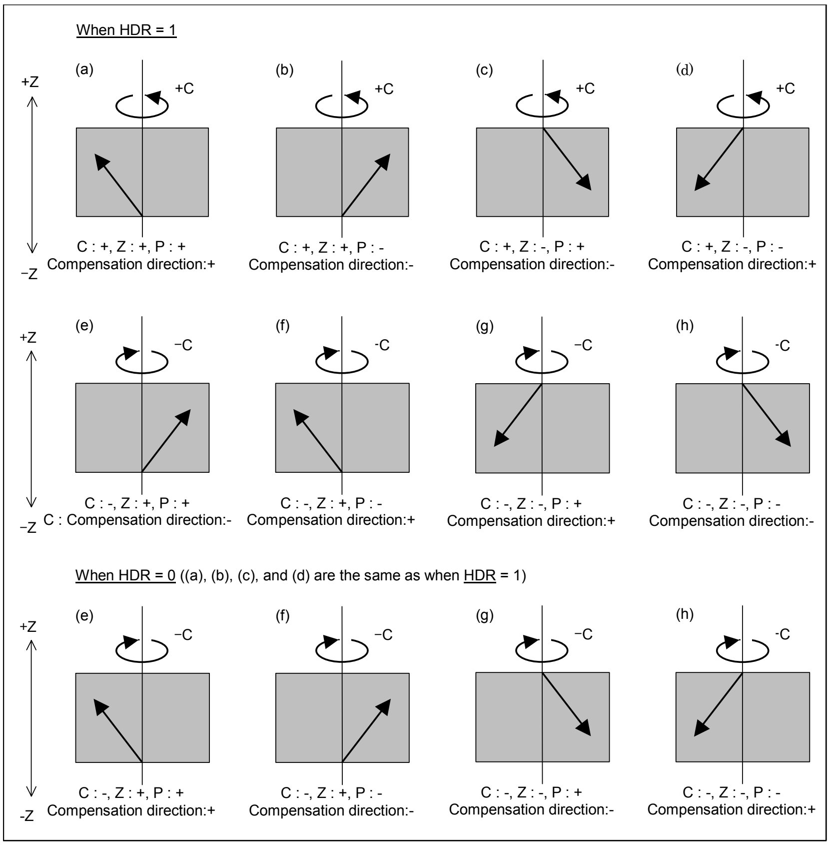
NOTE
1. Spindle-spindle polygon turning is enabled only for serial spindles.
2. This parameter is invalid if either parameter No. 7642 or No.7643 is set to 0. In this case, the settings of parameter No. 7640 and No.7641 are valid.
3. When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control option is required to specify an S command for the master axis.
4. When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
5. A spindle number common to the system is to be set in this parameter. When using this parameter, set 0 in parameter No. 7640 and No. 7641.

	#7	#6	#5	#4	#3	#2	#1	#0
7700						HDR		HBR

[Input type] Parameter input
[Data type] Bit path

 # 0 HBR When the electric gear box (EGB) function is used, performing a reset:
0: Cancels the synchronous mode (G81 or G81.5).
1: Does not cancel the synchronous mode. The mode is canceled only by the G80 or G80.5 command.

 # 2 HDR Direction for compensation for a helical gear (usually, set 1.)
(Example) To cut a left-twisted herical gear when the direction of rotation about the C-axis is the negative (-) direction:
0: Set a negative (-) value in P.
1: Set a positive (+) value in P.



	#7	#6	#5	#4	#3	#2	#1	#0
7701					**LZR**			

[Input type] Parameter input
[Data type] Bit path

 # 3 LZR When L (number of hob threads) = 0 is specified at the start of EGB synchronization (G81):
0: Synchronization is started, assuming that L = 1 is specified.
1: Synchronization is not started, assuming that L = 0 is specified. However, helical gear compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7702	**PHD**	**PHS**			**ART**			**TDP**

[Input type] Parameter input
[Data type] Bit path

 # 0 TDP The specifiable number of teeth, T, of the electric gear box (G81) is:
0: 1 to 1000
1: 0.1 to 100 (1/10 of a specified value)

> **NOTE**
> In either case, a value from 1 to 1000 can be specified.

 # 3 ART The retract function executed when a servo spindle alarm is issued is:
0: Disabled.
1: Enabled.

 # 6 PHS When the G81/G80 block contains no R command:
0: Acceleration/deceleration is not performed at the start or cancellation of EGB synchronization.
1: Acceleration/deceleration is performed at the start or cancellation of EGB synchronization. After acceleration at the start of synchronization, phase synchronization is automatically performed.

 # 7 PHD The direction of movement for automatic phase synchronization is:
0: Positive (+).
1: Negative (-).

	#7	#6	#5	#4	#3	#2	#1	#0
7703						ARO	ARE	ERV

[Input type] Parameter input
[Data type] Bit path

 # 0 ERV During EGB synchronization (G81), feed per revolution is performed for:
0: Feedback pulses.
1: Pulses converted to the speed for the workpiece axis.

 # 1 ARE The retract function executed when a servo spindle alarm is issued retracts the tool during:
0: EGB synchronization or automatic operation (automatic operation signal OP = 1).
1: EGB synchronization.

 # 2 ARO The retract function executed when a servo spindle alarm is issued retracts the tool during:
0: EGB synchronization.
1: EGB synchronization and automatic operation (automatic operation signal OP = 1).

The following table lists the parameter settings and corresponding operation.

ARE	ARO	Operation
1	0	During EGB synchronization
1	1	During EGB synchronization and automatic operation
0	0	During EGB synchronization or automatic operation
0	1	

> **NOTE**
> 1 Parameters ARE and ARO are valid when bit 3 (ART) of parameter No. 7702 is set to 1 (when the retract function executed when a servo spindle alarm is issued is enabled).
> 2 This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to 1.

7710	Axis number of an axis to be synchronized using the method of command specification for a hobbing machine

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes

When there are several groups of axes to be synchronized (the axes for which bit 0 (SYNMOD) of parameter No. 2011 is set to 1), an axis for which to start synchronization is specified using the following command (for a hobbing machine):

G81 T t L ±*l* ;

t: Spindle speed ($1 \le t \le 1000$)

l: Number of synchronized axis rotations ($1 \le l \le 21$)

Synchronization between the spindle and a specified axis is established with the ratio of ±*l* rotations about the synchronized axis to t spindle rotations.
t and *l* correspond to the number of teeth and the number of threads on the hobbing machine, respectively.
When there are several groups of axes to be synchronized and the above command is issued without setting this parameter, the alarm (PS1593) is issued.
When only one group of axes is to be synchronized, this parameter is ignored.

	#7	#6	#5	#4	#3	#2	#1	#0
7731					ECN			EFX

[Input type] Parameter input
[Data type] Bit path

 # 0 EFX As the EGB command:
0: G80 and G81 are used.
1: G80.8 and G81.8 are used.

> **NOTE**
> When this parameter is set to 0, no drilling canned cycle can be used.

 # 3 ECN When the automatic phase synchronization function for the electric gear box is disabled, during EGB synchronization, the G81 or G81.5 command:

0: Cannot be issued again. (The alarm (PS1595) is issued.)

1: Can be issued again.

7740	Feedrate during retraction

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

This parameter sets the feedrate during retraction for each axis.

7772	Number of position detector pulses per rotation about the tool axis

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999

This parameter sets the number of pulses per rotaiton about the tool axis (on the spindle side), for the position detector.

For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7773	Number of position detector pulses per rotation about the workpiece axis

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999

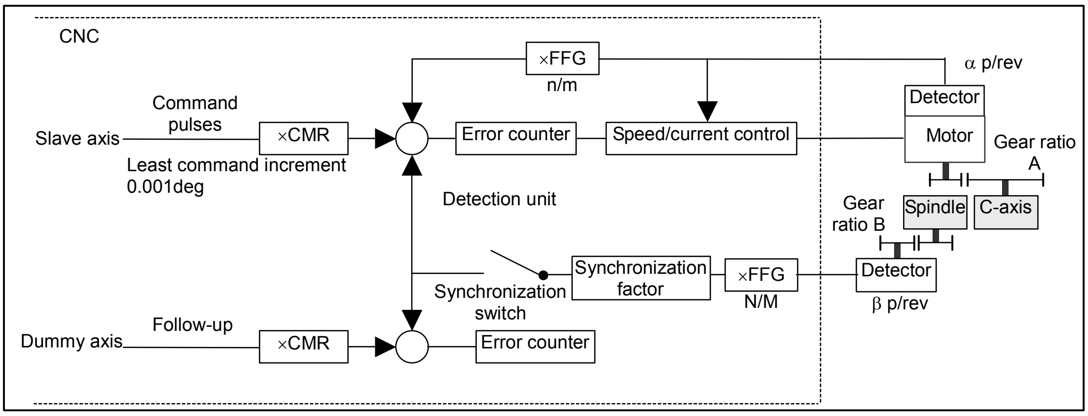
This parameter sets the number of pulses per rotation about the workpiece axis (on the slave side), for the position detector.

Set the number of pulses output by the detection unit.

Set parameters Nos. 7772 and 7773 when using the G81 EGB synchronization command.

(Example 1)

When the EGB master axis is the spindle and the EGB slave axis is the C-axis



Gear ratio of the spindle to the detector B:
1/1 (The spindle and detector are directly connected to each other.)
Number of detector pulses per spindle rotation β: 80,000 pulses/rev
(Calculated for four pulses for one A/B phase cycle)
FFG N/M of the EGB dummy axis: 1/1
Gear ratio of the C-axis A: 1/36 (One rotation about the C-axis to 36 motor rotations)
Number of detector pulses per C-axis rotation α: 1,000,000 pulses/rev
C-axis CMR: 1
C-axis FFG n/m: 1/100

In this case, the number of pulses per spindle rotation is:
$80000 \times 1/1 = 80000$
Therefore, set 80000 for parameter No. 7772.
The number of pulses per C-axis rotation in the detection unit is:
$1000000 \div 1/36 \times 1/100 = 360000$
Therefore, set 360000 for parameter No. 7773.

(Example 2)
When the gear ratio of the spindle to the detector B is 2/3 for the above example (When the detector rotates twice for three spindle rotations)
In this case, the number of pulses per spindle rotation is:

$$80000 \times \frac{2}{3} = \frac{160000}{3}$$

160000 cannot be divided by 3 without a remainder. In this case, change the setting of parameter No. 7773 so that the ratio of the settings of parameters Nos. 7772 and 7773 indicates the value you want to set.

$$\frac{\text{No.5996}}{\text{No.5997}} = \frac{160000/3}{360000} = \frac{160000}{360000 \times 3} = \frac{160000}{1080000}$$

Therefore, set 160000 for parameter No. 7772 and 1080000 for parameter No. 7773.
As described above, all the settings of parameters Nos. 7772 and 7773 have to do is to indicate the ratio correctly. So, you can reduce the fraction indicated by the settings. For example, you may set 16 for parameter No. 7772 and 108 for parameter No. 7773 for this example.

7776	Feedrate during automatic phase synchronization for the workpiece axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg/min
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
This parameter sets the feedrate during automatic phase synchronization for the workpiece axis.
When this parameter is set to 0, the rapid traverse rate (parameter No. 1420) is used as the feedrate during automatic phase synchronization.

7777	Angle shifted from the spindle position (one-rotation signal position) the workpiece axis uses as the reference of phase synchronization

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the angle shifted from the spindle position (one-rotation signal position) the workpiece axis uses as the reference of phase synchronization.

7778	Acceleration for acceleration/deceleration for the workpiece axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/sec/sec
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(For a millimeter machine, 0.0 to +100000.0, for an inch machine, 0.0 to +10000.0)
This parameter sets an acceleration for acceleration/deceleration for the workpiece axis.

7782	Number of pulses from the position detector per EGB master axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Valid data range] 1 to 999999999
This parameter sets the number of pulses from the position detector per EGB master axis rotation.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7783	Number of pulses from the position detector per EGB slave axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999
This parameter sets the number of pulses from the position detector per EGB slave axis rotation.
Set the number of pulses output by the detection unit.
Set this parameter when using the G81.5 EGB synchronization command.
The method for setting parameters Nos. 7782 and 7783 is the same as for parameters Nos. 7772 and 7773. For the method, see the description of parameters Nos. 7772 and 7773.

The ratio of the number of pulses for the master slave to that of pulses for the slave axis may be valid, but the settings of the parameters may not indicate the actual number of pulses. For example, the number of pulses may not be able to be divided without a remainder for the reason of the master and slave axis gear ratios as described in example 2. In this case, the following methods cannot be used for the G81.5 command:

G81.5 T__ C__ ; When the speed is specified for the master axis and the travel distance is specified for the slave axis
G81.5 P__ C0 L__ ; When the number of pulses is specified for the master axis and the speed is specified for the slave axis

	#7	#6	#5	#4	#3	#2	#1	#0
8001					RDE	OVE		MLE

[Input type] Parameter input
[Data type] Bit path

 # 0 MLE Whether all axis machine lock signal MLK is valid for PMC-controlled axes
0: Valid
1: Invalid
The axis-by-axis machine lock signal MLKx depends on the setting of bit 1 of parameter No. 8006.

 # 2 OVE Signals related to dry run and override used in PMC axis control
0: Same signals as those used for the CNC
1: Signals specific to the PMC
The signals used depend on the settings of these parameter bits as indicated below.

Signals	No.8001#2=0 (same signals as those used for the CNC)		No.8001#2=1 (signals specific to the PMC)	
Feedrate override signals	*FV0to*FV7	G012	*EFV0to*EFV7	G151
Override cancellation signal	OVC	G006#4	EOVC	G150#5
Rapid traverse override signals	ROV1,2	G014#0,1	EROV1,2	G150#0,1
Dry run signal	DRN	G46#7	EDRN	G150#7
Rapid traverse selection signal	RT	G19#7	ERT	G150#6

(The signal addresses at PMC selection time are for the first group.)

 # 3 RDE Whether dry run is valid for rapid traverse in PMC axis control
0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10			RPD

[Input type] Parameter input
[Data type] Bit path

 # 0 RPD Rapid traverse rate for PMC-controlled axes
0: Feedrate specified with parameter No.1420
1: Feedrate specified with the feedrate data in an axis control command by PMC

 # 3 F10 Least increment for the feedrate for cutting feed (per minute) in PMC axis control
The following settings are applied when bit 4 (PF1) of parameter No. 8002 is set to 0 and bit 5 (PF2) of parameter No. 8002 is set to 0.

	F10	IS-A	IS-B	IS-C	IS-D	IS-E
Millimeter input (mm/min)	0	10	1	0.1	0.01	0.001
	1	100	10	1	0.1	0.01
Inch input (inch/min)	0	0.1	0.01	0.001	0.0001	0.00001
	1	1	0.1	0.01	0.001	0.0001

 # 4 PF1
 # 5 PF2 Set the feedrate unit of cutting feedrate (feed per minute) for an axis controlled by the PMC.

P8002#5 PF2	P8002#4 PF1	Feedrate unit
0	0	1 / 1
0	1	1 / 10
1	0	1 / 100
1	1	1 / 1000

 # 6 FR1
 # 7 FR2 Set the feedrate unit for cutting feedrate (feed per rotation) for an axis controlled by the PMC.

P8002#7 FR2	P8002#6 FR1	Millimeter input (mm/rev)	Inch input (inch/rev)
0	0	0.0001	0.000001
1	1		
0	1	0.001	0.00001
1	0	0.01	0.0001

	#7	#6	#5	#4	#3	#2	#1	#0
8004		NCI						

[Input type] Parameter input
[Data type] Bit path

 # 6 NCI In axis control by the PMC, a position check at the time of deceleration is:
0: Performed.
1: Not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
8005								EDC

[Input type] Setting input
[Data type] Bit path

 # 0 EDC In axis control by the PMC, an external deceleration function is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
8006		EZR		EFD				

[Input type] Parameter input
[Data type] Bit path

 # 4 EFD When cutting feed (feed per minute) is used in PMC axis control, the specification unit of feedrate data is:
0: Unchanged (1 times).
1: 100 times greater.

> **NOTE**
> When this parameter is set to 1, bit 3 of parameter No. 8002 is invalid.

 # 6 EZR In PMC axis control, bit 0 (ZRNx) of parameter No. 1005 is:
0: Invalid.
With a PMC controlled axis, the alarm (PS0224) is not issued.
1: Valid.
A reference position return state check is made on a PMC controlled axis as with an NC axis according to the setting of bit 0 (ZRNx) of parameter No. 1005.

	#7	#6	#5	#4	#3	#2	#1	#0
8008								EMRx

[Input type] Parameter input
[Data type] Bit axis

 # 0 EMRx When a PMC axis control command is issued in mirror image state, the mirror image is:
0: Not considered.
1: Considered.
This parameter is valid in the mirror image mode set with the mirror image signals MI1 to MI8 (G106#0 to 7) set to 1 or bit 0 (MIRx) of parameter No. 12 set to 1.
If a movement is made along the same axis by doubly specifying a command with the CNC and PMC axis control when this parameter is set to 0, and the mirror image mode is set, a coordinate shift can occur afterwards. So, do not attempt to make such a movement.

8010	Selection of the DI/DO group for each axis controlled by the PMC

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 1 to 40

Specify the DI/DO group to be used to specify a command for each PMC-controlled axis.
For addresses of the fifth group and up, 1000 is added in steps of 4 groups.
For example:
The start address of the 10th group is G2154.
The start address of the 25th axis is G6142.

P8010	Description
1	DI/DO 1st group (G142 to G153) is used.
2	DI/DO 2nd group (G154 to G165) is used.
3	DI/DO 3rd group (G166 to G177) is used.
4	DI/DO 4th group (G178 to G189) is used.
5	DI/DO 5th group (G1142 to G1153) is used.
6	DI/DO 6th group (G1154 to G1165) is used.
:	:
13	DI/DO 13th group (G3142 to G3153) is used.
:	:
20	DI/DO 20th group (G4178 to G4189) is used.
21	DI/DO 21st group (G5142toG5153) is used.
:	:
29	DI/DO 29th group (G7142toG7153) is used.
:	:
35	DI/DO 35th group (G8166toG8177) is used.
36	DI/DO 36th group (G8178toG8189) is used.
37	DI/DO 37th group (G9142toG9153) is used.
38	DI/DO 38th group (G9154toG9165) is used.
39	DI/DO 39th group (G9166toG9177) is used.
40	DI/DO 40th group (G9178toG9189) is used.

NOTE
When a value other than the above is set, the axis is not controlled by the PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
8011								XRT

[Input type] Parameter input
[Data type] Bit axis

 # 0 XRT The axis that uses the group specified by parameter No. 8010 is:
0: Not controlled by the real time custom macro.
1: Controlled by the real time custom macro.

NOTE
1 This parameter is invalid for the axis for which 0 or a value outside the range is set by parameter No. 8010
2 When multiple axes are assigned to the same group by parameter No. 8010, these axes cannot be controlled by the real time custom macro. When multiple axes are assigned to the same group, be sure to set this bit to 0.
3 When this parameter (No. 8011) is all 0s, the axis is used for PMC axis control.

8030	Time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000

For each axis, this parameter sets a time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control.

NOTE
When 0 is set in this parameter, the value set in parameter No. 1622 is used.
The value set in parameter No. 1622 is used also for linear acceleration/deceleration after cutting interpolation.

	#7	#6	#5	#4	#3	#2	#1	#0
8103							**MWP**	

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 1 MWP To specify a P command for the waiting M code/balance cut:
0: A binary value is used as conventionally done.
1: A path number combination is used.

8110	**Waiting M code range (minimum value)**

8111	**Waiting M code range (maximum value)**

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 ,100to99999999

A range of M code values can be set by specifying a minimum waiting M coder value (parameter No. 8110) and a maximum waiting M code value (parameter No. 8111).
(parameter No. 8110) ≤ (waiting M code) ≤ (parameter No. 8111)
Set 0 in these parameters when the waiting M code is not used.

	#7	#6	#5	#4	#3	#2	#1	#0
8162						**PKUx**		

[Input type] Parameter input
[Data type] Bit axis

 # 2 PKUx In the parking state,
0: The absolute, relative, and machine coordinates are not updated.
1: The absolute and relative coordinates are updated. The machine coordinates are not updated.

> **NOTE**
> 1 With an axis for which polar coordinate interpolation is specified, set this parameter to 1. If this parameter is set to 0, a coordinate shift can occur when a single block stop or feed hold is performed in the polar coordinate interpolation mode.
> 2 With an axis that is set to function as a synchronous master axis and synchronous slave axis at the same time (with bit 1 (SYWx) of parameter No. 8167), set this parameter to 1.
> 3 With an axis specified in the three-dimensional coordinate conversion mode, set this parameter to 1. If this parameter is set to 0, the alarm (PS0367) is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8163	NUMx							

[Input type] Parameter input
[Data type] Bit axis

 # 7 NUMx When neither synchronous control nor composite control is applied, a move command for the axis is:
0: Not disabled.
1: Disabled.

> **NOTE**
> If a move command is specified for an axis with NUMx set to 1 when neither synchronous control nor composite control is applied, alarm PS0353 is issued.

8180	Master axis with which an axis is synchronized under synchronous control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . . , 1001, 1002, 1003, . . .)

This parameter sets the path number and intra-path relative axis number of the master axis with which each axis is synchronized. When zero is specified, the axis does not become a slave axis and is not synchronized with another axis. When an identical number is specified in two or more parameters, one master axis has two or more slave axes.

8183	Composite control axis of the other path in composite control for each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . . , 1001, 1002, 1003, . . .)

This parameter sets with which axis each axis is to be placed under composite control. When zero is specified, control of the axis is not replaced under composite control. An identical number can be specified in two or more parameters, but composite control cannot be exercised for all of tem at a time.

NOTE
When using the two-path interface, set this parameter for path 2.

8186	Master axis under superimposed control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . . , 1001, 1002, 1003, . . .)

This parameter sets the path number and intra-path relative axis number of a superimposed master axis for each axis when superimposed control is exercised. When zero is specified, the axis does not become a slave axis under superimposed control and the move pulse of another axis is not superimposed.

An identical number can be specified in two or more parameters to exercise superimposed control simultaneously. This means that superimposed control with one master axis and multiple slave axes is possible.

A slave axis may function as the master axis of another axis to allow three-generation superimposed control: parent (master axis) - child (slave axis/master axis) - grandchild (slave axis).

In this case, a movement along the child is made by its travel distance plus the travel distance of the parent, and a movement along the grandchild is made by its travel distance plus the travel distance of the child plus the travel distance of the parent.

Example of the relationship of parent (X1 of path 1) - child (X2 of path 2) - grandchild (X3 of path 3):
The travel distance of X1 is superimposed on X2, and the travel distances of X1 and X2 are further superimposed on X3.
Parameter No. 8186x of path 2 = 101
Parameter No. 8186x of path 3 = 201

	#7	#6	#5	#4	#3	#2	#1	#0
8200						AZR		AAC

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 AAC

0: Does not perform angular axis control.
1: Performs inclined axis control.

 # 2 AZR

0: The machine tool is moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.
1: The machine tool is not moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.

	#7	#6	#5	#4	#3	#2	#1	#0
8201		A53				AO3	AO2	AOT

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 AOT Stored stroke limit 1 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

 # 1 AO2 Stored stroke limit 2 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

 # 2 AO3 Stored stroke limit 3 under angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

 # 6 A53 So far, if a slanted axis is singly specified by a machine coordinate command (G53) in angular axis control, this parameter set to 0 specifies that "compensation is applied to the Cartesian axis", and this parameter set to 1 specifies that "a movement is made along the slanted axis only". However, the specification has been changed so that "a movement is made along the slanted axis only", regardless of whether this parameter is set to 0 or 1.

8210	Slant angle of a slanted axis in angular axis control

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] -180.000 to 180.000. However, angular axis control is disabled in the ranges -95.000 to -85.000 and 85.000 to 95.000 (in the case of IS-B).

8211	Axis number of a slanted axis subject to angular axis control

8212	Axis number of a Cartesian axis subject to slanted axis control

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to number of controlled axes

When angular axis control is to be applied to an arbitrary axis, these parameters set the axis numbers of a slanted axis and Cartesian axis. If 0 is set in either of the two parameters, the same number is set in the two parameters, or a number other than the controlled axis numbers is set in either of the two parameters, a slanted axis and Cartesian axis are selected as indicated in the following table:

	Slanted axis	Cartesian axis
M series	Y-axis (axis with 2 set in parameter No. 1022) of the basic three axes	Z-axis (axis with 3 set in parameter No. 1022) of the basic three axes
T series	X-axis (axis with 1 set in parameter No. 1022) of the basic three axes	Z-axis (axis with 3 set in parameter No. 1022) of the basic three axes

	#7	#6	#5	#4	#3	#2	#1	#0
8301				SYA				

[Input type] Parameter input
[Data type] Bit path

 # 4 SYA In the servo-off state in feed axis synchronous control, the limit of the difference between the positioning deviation of the master axis and that of the slave axis is:
0: Not checked.
1: Checked.

	#7	#6	#5	#4	#3	#2	#1	#0
8302	SMA							

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 7 SMA When an absolute position detector is attached, and bit 4 (APZ) of parameter No. 1815 for an axis in synchronous operation is set to OFF, APZ of the pairing axis in synchronous operation is:
0: Not set to OFF.
1: Set to OFF.

	#7	#6	#5	#4	#3	#2	#1	#0
8303	SOF					SAF	ATS	ATE

[Input type] Parameter input
[Data type] Bit axis

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 ATE In feed axis synchronous control, automatic setting for grid positioning is:
0: Disabled
1: Enabled
Set this parameter with a slave axis.

 # 1 ATS In feed axis synchronous control, automatic setting for grid positioning is:
0: Not started
1: Started
Set this parameter with a slave axis.

> **NOTE**
> When starting automatic setting for grid positioning, set ATS to 1. Upon the completion of setting, ATS is automatically set to 0.

 # 2 SAF In feed axis synchronous control, a movement along a slave axis is:
0: Not added to actual feedrate display.
1: Added to actual feedrate display.
Set this parameter with a slave axis.

 # 7 SOF In feed axis synchronous control, the synchronization establishment function based on machine coordinates is:
0: Disabled.
1: Enabled.
Set this parameter with a slave axis.
When using synchronization error compensation, set this parameter to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
8304	SYE	SMS	SCA	MVB	CLP	ADJ		SSA

[Input type] Parameter input
[Data type] Bit axis

 # 0 SSA When the one-direction synchronization establishment function under feed axis synchronous control is used:
0: The axis with a larger machine coordinate is used as the reference.
1: The axis with a smaller machine coordinate is used as the reference.

> **NOTE**
> 1 When this parameter is set, the power must be turned off before operation is continued.
> 2 Set this parameter (SSA) to the same value for both the master and slave axes.

 # 2 ADJ In feed axis synchronous control, this parameter specifies an axis along which a movement is made in the modification mode.
0: A movement is not made in the modification mode along the axis.
1: A movement is made in the modification mode along the axis.
When this parameter is set to 1, the modification mode is set.
Along an axis with this parameter set to 1, a movement is made by a move command for the master axis.

Set this parameter for one of the master and slave axes.
When there are multiple slave axes for one master axis, set this parameter to 1 for an axis with which a synchronization error excessive alarm is issued for recovery. If an alarm is issued with multiple axes, modify this parameter after recovery of one axis to recover another axis.

 # 3 CLP In axis feed synchronous control, synchronization error compensation is:
0: Disabled.
1: Enabled.
Set this parameter with a slave axis.

 # 4 MVB In the modification mode, a move command in a direction that increases a synchronization error is:
0: Ignored.
1: Valid.
When there are multiple slave axes for one master axis, an attempt to reduce the synchronous error of a slave axis by a movement along the master axis can increase the synchronization error of another slave axis. If this parameter is set to 0 in such a case, a movement can be made in neither direction along the master axis. In this case, set bit 2 (ADJ) of parameter No. 8304 to make a movement along a slave axis to perform a corrective operation.

 # 5 SCA In feed axis synchronous control:
0: Synchronous operation is performed when the feed axis synchronous control manual feed selection signal SYNCJ or the feed axis synchronous control selection signal SYNC for slave axes is set to 1.
1: Synchronous operation is performed at all times.
Set this parameter with a slave axis.

 # 6 SMS The synchronization error smooth suppress function is:
0: Disabled.
1: Enabled.
Set this parameter with a slave axis.

 # 7 SYE When external machine coordinate system shift is specified by external data input/output for the master axis in synchronous control, the slave axis is:
0: Not shifted.
1: Shifted by the same amount as specified for the master axis.
Set this parameter for the slave axis.
This function is disabled during normal operation.

	#7	#6	#5	#4	#3	#2	#1	#0
8305							SSE	SSO

[Input type] Parameter input
[Data type] Bit path

 # 0 SSO The uni-directional synchronization function in feed axis synchronous control is:
0: Disabled.
1: Enabled.

 # 1 SSE After emergency stop, the uni-directional synchronization function in feed axis synchronous control is:
0: Disabled.
1: Enabled.

8311	Axis number of master axis in feed axis synchronous control

> **NOTE**
> Set this parameter to the same value for both the master and slave axes.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to Number of controlled axes

Select a master axis in feed axis synchronous control. In the parameter for the slave axis, set the axis number of the master axis.

Example 1)
When one set of feed axis synchronous control is used:
When the master axis is the first axis (X-axis), and the slave axis is the third axis (Z-axis), set parameter No. 8311 as follows:
Parameter No.8311 X (first axis) = 0
Parameter No.8311 Y (second axis) = 0
Parameter No.8311 Z (third axis) = 1
Parameter No.8311 A (fourth axis) = 0

Example 2)
When three sets of feed axis synchronous control is used:
When the master axes are the first axis, second axis, and third axis, and the slave axes are the sixth axis, fifth axis, and fourth axis, set parameter No. 8311 as follows:
Parameter No.8311 X (first axis) = 0
Parameter No.8311 Y (second axis) = 0
Parameter No.8311 Z (third axis) = 0
Parameter No.8311 A (fourth axis) = 3
Parameter No.8311 B (fifth axis) = 2
Parameter No.8311 C (sixth axis) = 1

8312	Enabling/disabling mirror image in feed axis synchronous control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] -127 to 128

This parameter sets mirror image for the slave axis. When 100 or a more value is set with this parameter, the mirror image function is applied to synchronous control. Set this parameter to the slave axis.

Example)

For reverse synchronization with the master axis being the third axis and the slave axis being the fourth axis, set parameter No. 8312 as follows:
Parameter No.8312 X (first axis) = 0
Parameter No.8312 Y (second axis) = 0
Parameter No.8312 Z (third axis) = 0
Parameter No.8312 A (fourth axis) = 100

NOTE
In synchronous operation with mirror image applied, synchronization error compensation, synchronization establishment, synchronization error checking, and modification mode cannot be used.

8314	Maximum allowable error in synchronization error check based on machine coordinates

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets a maximum allowable error in a synchronization error check based on machine coordinates. When the error between the master and slave axes in machine coordinates exceeds the value set in this parameter, the machine stops with the servo alarm (SV0005).
Set this parameter with a slave axis.

NOTE
Set 0 in this parameter when a synchronization error check is not made.

8323	Limit in positional deviation check in feed axis synchronous control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

This parameter sets the maximum allowable difference between the master axis and slave axis position deviations. When the absolute value of a positional deviation difference exceeds the value set in this parameter in feed axis synchronous control, the alarm (DS0001) is issued.
Set this parameter with a slave axis. If 0 is specified in this parameter, no position deviation difference check is made.

8325	Maximum compensation value in synchronization establishment based on machine coordinates

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the maximum compensation value for synchronization. When a compensation value exceeding the value set in this parameter is detected, the servo alarm (SV0001) is issued, and the synchronization establishment is not performed.
Specify a slave axis for this parameter. To enable this parameter, set the parameter SOF (bit 7 of parameter No.8303) to 1. When 0 is set in this parameter, synchronization establishment is not performed.

8326	Difference between master axis and slave axis reference counters

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

The difference between the master axis reference counter and slave axis reference counter (master axis and slave axis grid shift) is automatically set when automatic setting for grid positioning is performed. Then, the difference is transferred together with an ordinary grid shift value to the servo system when the power is turned on. This parameter is set with a slave axis.

8327	Torque difference alarm detection timer

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000

This parameter sets a time from the servo preparation completion signal, SA (F000#6), being set to 1 until torque difference alarm detection is started in feed axis synchronous control.
When 0 is set in this parameter, the specification of 512 msec is assumed.
Set this parameter with a slave axis.

8330	Multiplier for a maximum allowable synchronization error immediately after power-up

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 100

Until synchronization establishment is completed immediately after power-up, synchronization error excessive alarm 2 is checked using the maximum allowable error (parameter No. 8332) multiplied by the value set in this parameter.
If the result produced by multiplying the value of parameter No. 8332 by the value of this parameter exceeds 32767, the value is clamped to 32767.

8331	Maximum allowable synchronization error for synchronization error excessive alarm 1

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767

This parameter sets a maximum allowable synchronization error for synchronization error excessive alarm 1.
Set this parameter with a slave axis.

8332	Maximum allowable synchronization error for synchronization error excessive alarm 2

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767

This parameter sets a maximum allowable synchronization error for synchronization error excessive alarm 2.
Set this parameter with a slave axis.

8333	Synchronization error zero width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767

When a synchronization error below the value set in this parameter is detected, synchronization error compensation is not performed.
Set this parameter with a slave axis.

8334	Synchronization error compensation gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 1 to 1024

This parameter sets a synchronization error compensation gain.
Compensation pulses found by the following expression are output for the slave axis:

Compensation pulses = Synchronization error × (Ci/1024)
Ci: Compensation gain

Set this parameter with a slave axis.

8335	Synchronization error zero width 2 for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter sets synchronization error zero width 2 for synchronization error smooth suppression.
Set this parameter with a slave axis.

NOTE
Set a value less than the value set in parameter No. 8333.

8336	Synchronization error compensation gain 2 for each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1024

This parameter sets synchronization error compensation gain 2 for synchronization error smooth suppression.
Set this parameter with a slave axis.

> **NOTE**
> Set a value less than the value set in parameter No. 8334.

8337	M code for turning off synchronization in feed axis synchronous control

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999

This parameter specifies an M code for switching from synchronous operation to normal operation.
The M code set in this parameter is not buffered.

8338	M code for turning on synchronization in feed axis synchronous control

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999

This parameter specifies an M code for switching from normal operation to synchronous operation.
The M code set in this parameter is not buffered.

	#7	#6	#5	#4	#3	#2	#1	#0
8451	NOF			ZAG				

[Input type] Setting input
[Data type] Bit path

 # 4 ZAG The deceleration function based on cutting load in AI contour control (deceleration based on Z-axis fall angle) is:
0: Not performed.
1: Performed.
When this parameter is set to 1, be sure to set parameter Nos. 8456, 8457, and 8458.

 # 7 NOF In AI contour control, an F command is:
0: Not ignored.
1: Ignored.
When this parameter is set to 1, the specification of the maximum allowable feedrate set in parameter No. 8465 is assumed.

8456	Override for range 2 that is applied during deceleration according to the cutting load in AI contour control

8457	Override for range 3 that is applied during deceleration according to the cutting load in AI contour control

8458	Override for range 4 that is applied during deceleration according to the cutting load in AI contour control

[Input type] Setting input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 100

For the function of decelerating according to the cutting load in AI contour control, the override set in a parameter can be applied according to the angle at which the tool moves downward along the Z-axis. The feedrate obtained according to other conditions is multiplied by the override for the range containing angle θ at which the tool moves downward.

However, when bit 1 (ZG2) of parameter No. 19515 is set to 0, no parameter is available to range 1, and 100% is applied at all times. When bit 1 (ZG2) of parameter No. 19515 is set to 1, set an override value for range 1 in parameter No. 19516.

Range 1 $0° \le \theta < 30°$
Range 2 $30° \le \theta < 45°$
Range 3 $45° \le \theta < 60°$
Range 4 $60° \le \theta \le 90°$

8465	Maximum allowable feedrate for AI contour control

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

This parameter sets the maximum allowable feedrate for contour control.

If a feedrate higher than the setting of this parameter is specified in the AI contour control mode, the feedrate is clamped to that set in this parameter.

If this parameter is set to 0, no clamping is performed.

When bit 7 (NOF) of parameter No. 8451 is set to 1, the tool moves, assuming that the feedrate set in this parameter is specified. If 0 is set in this parameter at this time, a movement is made at the specified feedrate.

8486	Maximum travel distance of a block where smooth interpolation or Nano smoothing is applied

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter specifies a block length used as a reference to decide whether to apply smooth interpolation or Nano smoothing. If the line specified in a block is longer than the value set in the parameter, smooth interpolation will not be applied to that block.

8487	Angle at which smooth interpolation or Nano smoothing is turned off

[Input type] Setting input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90

This parameter sets the angle used to determine whether to apply smooth interpolation or Nano smoothing.
At a point having a difference in angle greater than this setting, smooth interpolation or Nano smoothing is turned off.

8490	Minimum travel distance of a block where smooth interpolation or Nano smoothing is applied

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a block length used to determine whether to apply smooth interpolation or Nano smoothing.
If the line specified in a block is shorter than the value set in this parameter, smooth interpolation or Nano smoothing is not applied to that block.

	#7	#6	#5	#4	#3	#2	#1	#0
8900								PWE

[Input type] Setting input
[Data type] Bit

 # 0 PWE The setting, from an external device and MDI panel, of those parameters that cannot be set by setting input is:
0: Disabled.
1: Enabled.

10461	RGB value of color palette 1 for text for color set 3
10462	RGB value of color palette 2 for text for color set 3
	:
10475	RGB value of color palette 15 for text for color set 3

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 151515

Each of these parameters sets the RGB value of each color palette for text by specifying a 6-digit number as described below.
rrggbb: 6-digit number (rr: red data, gg: green data, bb: blue data)
The valid data range of each color is 0 to 15 (same as the tone levels on the color setting screen). When a number equal to or greater than 16 is specified, the specification of 15 is assumed.
Example)
When the tone level of a color is: red:1 green:2, blue:3, set 10203 in the parameter.

10800	First compensation axis for three-dimensional error compensation
10801	Second compensation axis for three-dimensional error compensation
10802	Third compensation axis for three-dimensional error compensation

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

These parameters set three compensation axes for applying three-dimensional error compensation.

10803	Number of compensation points for three-dimensional error compensation (first compensation axis)
10804	Number of compensation points for three-dimensional error compensation (second compensation axis)
10805	Number of compensation points for three-dimensional error compensation (third compensation axis)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 2 to 25
These parameters set the number of compensation points for each axis for three-dimensional error compensation.

10806	Compensation point number of the reference position for three-dimensional error compensation (first compensation axis)
10807	Compensation point number of the reference position for three-dimensional error compensation (second compensation axis)
10808	Compensation point number of the reference position for three-dimensional error compensation (third compensation axis)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of compensation points
These parameters set the compensation point number of the reference position for each axis for three-dimensional error compensation.

10809	Magnification for three-dimensional error compensation (first compensation axis)
10810	Magnification for three-dimensional error compensation (second compensation axis)
10811	Magnification for three-dimensional error compensation (third compensation axis)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 100
These parameters set the magnification for each axis for three-dimensional error compensation.

10812	Compensation interval for three-dimensional error compensation (first compensation axis)
10813	Compensation interval for three-dimensional error compensation (second compensation axis)
10814	Compensation interval for three-dimensional error compensation (third compensation axis)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the compensation interval for each axis for three-dimensional error compensation.

12310	States of the first manual handle feed axis selection signals when tool axis direction handle feed/interrupt and table-based vertical direction handle feed/interrupt are performed

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the first manual handle feed axis selection signals (HS1A to HS1E)/manual handle interrupt axis selection signals (HS1IA to HS1IE) when tool axis direction handle feed/interrupt and table-based vertical direction handle feed/interrupt are performed.

<Table of correspondence with the manual handle feed axis selection signals>

The table below indicates the correspondence between the states of the first manual handle feed axis selection signals/manual handle interrupt axis selection signals and the parameter settings in the 5-axis machining manual feed (handle feed) mode. When the first manual handle pulse generator is turned after setting the signals corresponding to the value set in the parameter, operation is performed in the specified mode.

HS1E (HS1IE)	HS1D (HS1ID)	HS1C (HS1IC)	HS1B (HS1IB)	HS1A (HS1IA)	Parameter setting
0	0	0	0	1	1
0	0	0	1	0	2
0	0	0	1	1	3
0	0	1	0	0	4
0	0	1	0	1	5
0	0	1	1	0	6
0	0	1	1	1	7
0	1	0	0	0	8
0	1	0	0	1	9
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24

12311	States of the first manual handle feed axis selection signals when a movement is made in the first axis direction in tool axis normal direction handle feed/interrupt and table-based horizontal direction handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the first manual handle feed axis selection signals (HS1A to HS1E)/manual handle interrupt axis selection signals (HS1IA to HS1IE) when a movement is made in the first axis direction. (For a value to be set, see "Table of correspondence with the manual handle feed axis selection signals" provided in the description of parameter No. 12310.)

The table below indicates the relationships of tool axis directions, first axis directions, and second axis directions.

Parameter No.19697	Tool axis directions	First axis directions	Second axis directions
1	X	Y	Z
2	Y	Z	X
3	Z	X	Y

Note, however, that the table above indicates the directions applicable when the angles of all rotation axes are set to 0.
In tool axis direction/tool axis normal direction feed (not table-based), the directions indicated above assume that 0 is set in parameter No. 19698 and No. 19699. When a rotation axis has made a turn or a nonzero value is set in these parameters in tool axis direction/tool axis normal direction feed, the relevant directions are inclined accordingly.

12312	States of the first manual handle feed axis selection signals when a movement is made in the second axis direction in tool axis normal direction handle feed/interrupt and table-based horizontal direction handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the second manual handle feed axis selection signals (HS1A to HS1E)/manual handle interrupt axis selection signals (HS1IA to HS1IE) when a movement is made in the first axis direction. (For a value to be set, see "Table of correspondence with the manual handle feed axis selection signals" provided in the description of parameter No. 12310.)

12313	States of the first manual handle feed axis selection signals when the first rotation axis is turned in tool tip center rotation handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the first manual handle feed axis selection signals (HS1A to HS1E)/manual handle interrupt axis selection signals (HS1IA to HS1IE) when the first rotation axis is turned in tool tip center rotation handle feed/interrupt. (For a value to be set, see "Table of correspondence with the manual handle feed axis selection signals" provided in the description of parameter No. 12310.)

12314	States of the first manual handle feed axis selection signals when the second rotation axis is turned in tool tip center rotation handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the first manual handle feed axis selection signals (HS1A to HS1E)/manual handle interrupt axis selection signals (HS1IA to HS1IE) when the second rotation axis is turned in tool tip center rotation handle feed/interrupt. (For a value to be set, see "Table of correspondence with the manual handle feed axis selection signals" provided in the description of parameter No. 12310.)

12318	Tool length in 5-axis machining manual feed

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a tool length when tool tip center rotation feed is performed with the 5-axis machining manual feed function and when the 5-axis machining manual feed screen is displayed.

NOTE
Specify a radius value to set this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
12320						JFR	FLL	TWD

[Input type] Parameter input
[Data type] Bit path

 # 0 TWD The directions of 5-axis machining manual feed (other than tool tip center rotation feed) when the tilted working plane command is issued are:

0: Same as those not in the tilted working plane command. That is, the directions are:
Tool axis normal direction 1 (table-based horizontal direciton 1)
Tool axis normal direction 2 (table-based horizontal direction 2)
Tool axis direction (table-based vertical direction)

1: X, Y, and Z directions in the feature coordinate system.

 # 1 FLL The directions of tool axis normal direction feed or table-based horizontal direction feed in the 5-axis machining manual feed mode are:

0: Tool axis normal direction 1 (table-based horizontal direction 1) and tool axis normal direction 2 (table-based horizontal direction 2).

1: Longitude direction and latitude direction.

Parameter FLL (No.12320#1)	Parameter TWD (No.12320#0)	Directions of 5-axis machining manual feed
0	0	Conventional directions
0	1	When the tilted working plane command is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Conventional directions
1	0	Longitude direction and latitude direction
1	1	When the tilted working plane command is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Longitude direction and latitude direction

 # 2 JFR As the feeedrate of 5-axis machining jog feed or incremental feed:

0: The dry run rate (parameter No. 1410) is used.

1: The jog feedrate (parameter No. 1423) is used.

12321	Normal axis direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

For longitude or latitude direction feed in the 5-axis machining manual feed mode, this parameter sets the axis parallel to the normal direction.

1 : Positive (+) X-axis direction
2 : Positive (+) Y-axis direction
3 : Positive (+) Z-axis direction
0 : Reference tool axis direction (parameter No. 19697)

12322	Angle used to determine whether to assume the tool axis direction to be parallel to the normal direction (parameter No. 12321)

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90

For latitude direction feed or longitude direction feed in the 5-axis machining manual feed mode, when the angle between the tool axis direction and normal direction (parameter No. 12321) is small, the tool axis direction is assumed to be parallel to the normal direciton (parameter No. 12321). This parameter sets the maximum angle at which the tool axis direction is assumed to be parallel to the normal direction.

When this parameter is set to 0 or a value outside the valid range, it is set to 1 degree.

	#7	#6	#5	#4	#3	#2	#1	#0
13113					CFD			CLR

[Input type] Parameter input
[Data type] Bit path

 # 0 CLR Upon reset, the display of a travel distance by 5-axis machining manual feed is:
0: Not cleared.
1: Cleared.

 # 3 CFD As feedrate F, the 5-axis machining manual feed screen displays:
0: Composite feedrate at the linear axis/rotation axis control point.
1: Feedrate at the tool tip.

	#7	#6	#5	#4	#3	#2	#1	#0
13200					ETE	TRT		

[Input type] Parameter input
[Data type] Bit path

 # 2 TRT As the remaining lifetime value for outputting the tool life arrival notice signal:
0: The remaining lifetime of the last tool is used.
1: The sum of the remaining lifetimes of the tools with the same type number is used.

> **NOTE**
> This parameter is valid when bit 3 (ETE) of parameter No. 13200 is set to 0 (arrival notice for each type number).

 # 3 ETE The tool life arrival notice signal is output:
0: For each tool type.
1: For each tool.

	#7	#6	#5	#4	#3	#2	#1	#0
13201							TDN	TDC

[Input type] Parameter input
[Data type] Bit system common

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 TDC The function of customizing the tool management data screen of the tool management function is:
0: Disabled.
1: Enabled.

 # 1 TDN On the screen dedicated to the tool management function, the state of tool life can be displayed using a character string:
0: Up to 6 characters long.
1: Up to 12 characters long.

	#7	#6	#5	#4	#3	#2	#1	#0
13202	DOM	DOT		DO2	DOB	DOY	DCR	

[Input type] Parameter input
[Data type] Bit

 # 1 DCR On the tool management function screen, tool nose radius compensation data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or combined system.

 # 2 DOY On the tool management function screen, Y-axis offset data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or combined system.

 # 3 DOB On the tool management function screen, B-axis offset data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or combined system.

 # 4 DO2 On the tool management function screen, the second geometry tool offset data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or combined system.

 # 6 DOT On the tool management function screen, the tool offset data (X, Z) of the T series is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or combined system.

 # 7 DOM On the tool management function screen, the tool offset data of the M series is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or combined system.

	#7	#6	#5	#4	#3	#2	#1	#0
13204								TDL

[Input type] Parameter input
[Data type] Bit system common

 # 0 TDL The protection function for tool management data using a key is:
0: Disabled.
1: Enabled.

13220	Number of valid tools in tool management data

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 64 (Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of valid tools in tool management data.

13222	Number of data items in the first cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64 (Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the first cartridge.

13223	Start pot number of the first cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999
This parameter sets the start pot number to be used with the first cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13227	Number of data items in the second cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the second cartridge.

13228	Start pot number of the second cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1to9999
This parameter sets the start pot number to be used with the second cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13232	Number of data items in the third cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the third cartridge.

13233	Start pot number of the third cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1to9999
This parameter sets the start pot number to be used with the third cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13237	Number of data items in the fourth cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the fourth cartridge.

13238	Start pot number of the fourth cartridge

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1to9999

This parameter sets the start pot number to be used with the fourth cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13252	M code for specifying a particular tool

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 65535

This parameter sets not a tool type number but an M code for directly specifying the T code of a particular tool.

13265	Number for selecting a spindle position offset number

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 999

This parameters sets an H/D code for selecting an offset number registered in the data of the tool attached at the spindle position.
When 0 is set, an ordinary used code such as H99/D99 is used. When a value other than 0 is set, H99/D99 no longer has a particular meaning. So, when H99/D99 is specified in this case, the specification of offset number 99 is assumed.
With the T series, address D only is used to specify a tool number and offset number, so that a restriction is imposed on the number of digits. So, the valid data range of this parameter varies according the number of digits of an offset number.
When the number of digits of an offset number is 1: to 9
When the number of digits of an offset number is 2: to 99
When the number of digits of an offset number is 3: to 999

	#7	#6	#5	#4	#3	#2	#1	#0
13600								MCR

[Input type] Parameter input
[Data type] Bit path

 # 0 MCR When an allowable acceleration rate adjustment is made with the machining condition selection function (machining parameter adjustment screen, precision level selection screen), parameter No. 1735 for the deceleration function based on acceleration in circular interpolation is:
0: Modified.
1: Not modified.

	#7	#6	#5	#4	#3	#2	#1	#0
13601								MPR

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 MPR The machining parameter adjustment screen is:
0: Displayed.
1: Not displayed.
Even when this parameter is set to 1, the precision level selection screen is displayed.

13610	**Acceleration rate for acceleration/deceleration before look-ahead interpolation in AI contour control (precision level 1)**

13611	**Acceleration rate for acceleration/deceleration before look-ahead interpolation in AI contour control (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)
Each of these parameters sets an acceleration rate for acceleration/ deceleration before interpolation in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13612	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 1)

13613	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Byte path
[Unit of data] msec
[Valid data range] 0 to 200

Each of these parameters sets an acceleration rate change time (bell-shaped) in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13614	Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration (precision level 1)

13615	Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration (precision level 10)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)

(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)

Each of these parameters sets an allowable acceleration rate change amount per 1 ms for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration during AI contour control.

Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13616	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations (precision level 1)**

13617	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)

Each of these parameters sets an allowable acceleration rate change amount per 1 ms for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations during AI contour control.

Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

NOTE

1 For an axis with 0 set in this parameter, parameter No. 13614 and No. 13615 (allowable acceleration rate change amount in speed control based on acceleration rate change under control on the rate of change of acceleration) are valid.

2 For an axis with 0 set in parameter No. 13614 and No. 13615 (allowable acceleration rate change amount in speed control based on acceleration rate change under control on the rate of change of acceleration), speed control based on acceleration rate change is disabled, so that the specification of this parameter has no effect.

13618	**Rate of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation when AI contour control is used (precision level 1)**

13619	**Rate of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation when AI contour control is used (precision level 10)**

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 50

Each of these parameters sets the rate (percentage) of the change time of the rate of change of acceleration to the change time of acceleration rate change in smooth bell-shaped acceleration/deceleration before look-ahead interpolation during AI contour control.
Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

NOTE
When 0 or a value not within the valid data range is set in this parameter, smooth bell-shaped acceleration/deceleration before look-ahead interpolation is not performed.

13620	**Allowable acceleration rate when AI contour control is used (precision level 1)**

13621	**Allowable acceleration rate when AI contour control is used (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)

Each of these parameters sets an allowable acceleration rate in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13622	Time constant for acceleration/deceleration after interpolation when AI contour control is used (precision level 1)

13623	Time constant for acceleration/deceleration after interpolation when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 1 to 512

Each of these parameters sets a time constant for acceleration/deceleration after interpolation when AI contour control is used. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13624	Corner speed difference when AI contour control is used (precision level 1)

13625	Corner speed difference when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

Each of these parameters sets an allowable speed difference for speed determination based on corner speed difference in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13626	Maximum cutting speed when AI contour control is used (precision level 1)

13627	Maximum cutting speed when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

Each of these parameters sets a maximum cutting speed in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13628	Parameter number corresponding to arbitrary item 1 when AI contour control is used

13629	Parameter number corresponding to arbitrary item 2 when AI contour control is used

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 65535
These parameters set the parameter numbers corresponding to arbitrary items 1 and 2.

NOTE
The parameter numbers corresponding to the following cannot be specified:
- Bit parameters
- Spindle parameters (No. 4000 to No. 4799)
- Parameters of real number type
- Parameters that require power-off (for which the alarm (PW0000) is issued)
- Nonexistent parameters

13630	Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 1 when AI contour control is used

13631	Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 2 when AI contour control is used

13632	Value with emphasis on speed (precision level 10) of the parameter corresponding to arbitrary item 1 when AI contour control is used

13633	Value with emphasis on speed (precision level 10) of the parameter corresponding to arbitrary item 2 when AI contour control is used

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Depend on the type of parameter for an arbitrary item
[Valid data range] Depend on the type of parameter for an arbitrary item
Each of these parameters sets a value with emphasis placed on speed or precision for a parameter.

14010	Maximum allowable travel distance when the reference position is established for a linear scale with an absolute address reference position

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets the maximum allowable travel distance at the FL feedrate when the reference position is established for a linear scale with an absolute address reference position. When the travel distance exceeds the setting of this parameter, the alarm (PS5326) (SCALE WITH REFERENCE POSITION: REFERENCE POSITION ESTABLISHMENT FAILED) is issued. When this parameter is set to 0, the maximum allowable travel distance is not checked.

14340	ATR value corresponding to slave 01 on FSSB line 1

14341	ATR value corresponding to slave 02 on FSSB line 1

:

14357	ATR value corresponding to slave 18 on FSSB line 1

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0to23,64,-56,-96

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 18 on FSSB line 1 (first optical connector).
The slave is a generic term for servo amplifiers and separate detector interface units connected via an FSSB optical cable to the CNC. Numbers 1 to 18 are assigned to slaves, with younger numbers sequentially assigned to slaves closer to the CNC.
A 2-axis amplifier consists of two slaves, and a 3-axis amplifier consists of three slaves. In each of these parameters, set a value as described below, depending on whether the slave is an amplifier, separate detector, or nonexistent.

- When the slave is an amplifier:
  Set a value obtained by subtracting 1 from the setting of parameter No. 1023 for the axis to which the amplifier is assigned.
- When the slave is a separate detector interface unit:
  Set 64 for the first separate detector interface unit (connected near the CNC), and set -56 for the second separate detector interface unit (connected far from the CNC).
- When the slave is nonexistent:
  Set -96.

NOTE

1 When the electric gear box (EGB) function is used Although an amplifier is not actually required for an EGB dummy axis, set this parameter with assuming that a dummy amplifier is connected. That is, as the address conversion table value for a nonexistent slave, set the value obtained by subtracting 1 from the setting of parameter No. 1023 for the EGB dummy axis, instead of -96.

2 When the FSSB is set to the automatic setting mode (when the parameter FMD (No.1902#0) is set to 0), parameter Nos. 14340 to 14357 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the parameter FMD (No.1902#0) is set to 1), be sure to directly set values in parameter Nos. 14340 to 14357.

Example of axis configuration and parameter settings

Example 1

CNC

Controlled axis number	Program axis name No.1020	Servo axis No.1023
1	X	1
2	Y	3
3	Z	4
4	A	2
5	B	5
6	C	6

	Slave number	ATR No.14340 to 14357	Axis
Single-axis amplifier	1	0	X
Two-axis amplifier	2	1	A
	3	2	Y
Two-axis amplifier	4	3	Z
	5	4	B
M1	6	64	(M1)
Single-axis amplifier	7	5	C
M2	8	-56	(M2)
	9 to 18	-96	(None)

CNC

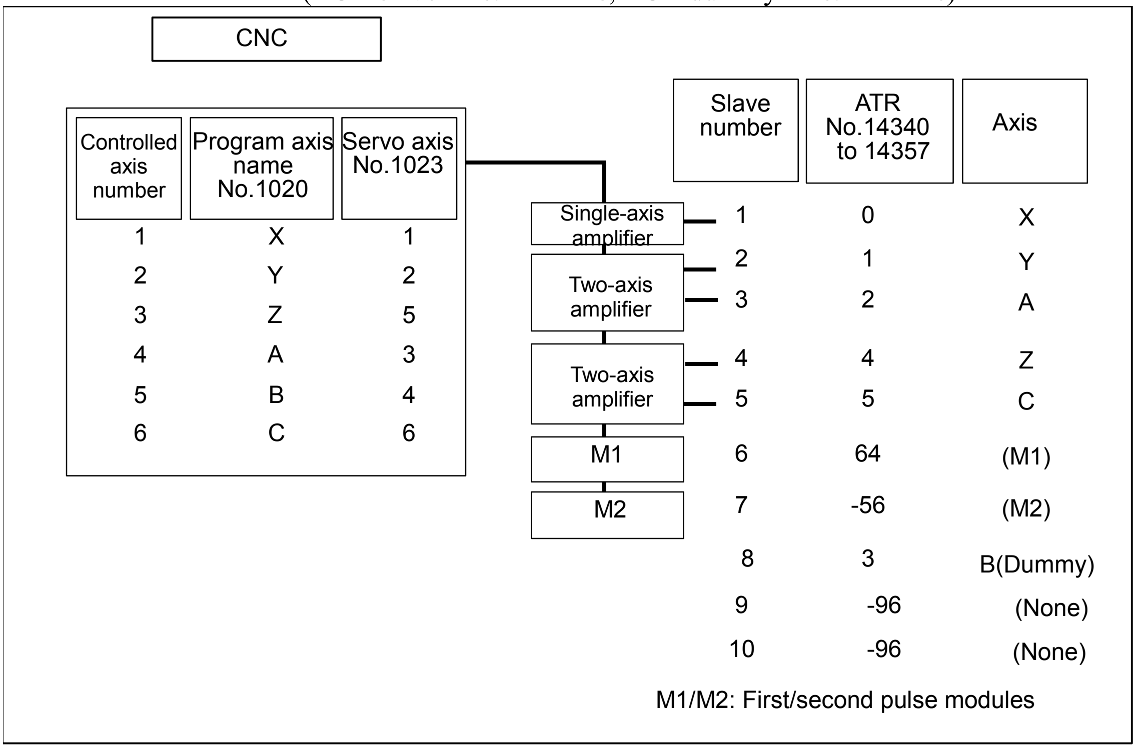
Controlled axis number	Program axis name No.1020	Servo axis No.1023
1	X	1
2	Y	3
3	Z	4
4	A	2
5	B	5
6	C	6

	Slave number	ATR No.14340 to 14357	Axis
Single-axis amplifier	1	0	X
Two-axis amplifier	2	2	Y
	3	3	Z
Two-axis amplifier	4	1	A
	5	4	B
M1	6	64	(M1)
Single-axis amplifier	7	5	C
M2	8	-56	(M2)
	9 to 18	-96	(None)

M1/M2: First/second pulse modules

Example 2

Example of axis configuration and parameter settings when the electric gear box (EGB) function is used
(EGB slave axis: A-axis, EGB dummy axis: B-axis)



14358	ASTR value corresponding to slave 01 on FSSB line 2
14359	ASTR value corresponding to slave 02 on FSSB line 2
:	
14375	ASTR value corresponding to slave 18 on FSSB line 2

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0to23,80,-40,-96

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 18 on FSSB line 2 (second optical connector).

- When the slave is an amplifier:
  Set a value obtained by subtracting 1 from the setting of parameter No. 1023 for the axis to which the amplifier is assigned.
- When the slave is a separate detector interface unit:
  Set 80 for the third separate detector interface unit (connected near the CNC), and set -40 for the fourth separate detector interface unit (connected far from the CNC).
- When the slave is nonexistent:
  Set -96.

NOTE
1 Set these parameters only when a servo axis control card with two optical connectors (FSSB lines) is used.
2 When the FSSB is set to the automatic setting mode (when the parameter FMD (No.1902#0) is set to 0), parameter Nos. 14358 to 14375 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the parameter FMD (No.1902#0) is set to 1), be sure to directly set values in parameter Nos. 14358 to 14375.

14376	**ATR value corresponding to connector 1 on the first separate detector interface unit**
14377	**ATR value corresponding to connector 2 on the first separate detector interface unit**
	:
14383	**ATR value corresponding to connector 8 on the first separate detector interface unit**
14384	**ATR value corresponding to connector 1 on the second separate detector interface unit**
	:
14391	**ATR value corresponding to connector 8 on the second separate detector interface unit**
14392	**ATR value corresponding to connector 1 on the third separate detector interface unit**
	:
14399	**ATR value corresponding to connector 8 on the third separate detector interface unit**
14400	**ATR value corresponding to connector 1 on the fourth separate detector interface unit**
	:
14407	**ATR value corresponding to connector 8 on the fourth separate detector interface unit**

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 32

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each connector on a separate detector interface unit.

The first and second separate detector interface units are connected to FSSB line 1, and the third and fourth separate detector interface units are connected to FSSB line 2.

In each of these parameters, set a value obtained by subtracting 1 from the setting of parameter No. 1023 for the axis connected to a connector on a separate detector interface unit.

When there is an axis for which bit 1 of parameter No. 1815 is set to 0 to use a separate detector interface unit, set 32 for those connectors that are not used.

> **NOTE**
> When the FSSB is set to the automatic setting mode (when the parameter FMD (No.1902#0) is set to 0), parameter Nos. 14376 to 14407 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the parameter FMD (No.1902#0) is set to 1), be sure to directly set values in parameter Nos. 14376 to 14407.

	#7	#6	#5	#4	#3	#2	#1	#0
19500		FNW						

[Input type] Parameter input
[Data type] Bit path

 # 6 FNW When the feedrate is determined according to the feedrate difference and acceleration in AI contour control:

0: The maximum feedrate at which the allowable feedrate difference and acceleration for each axis are not exceeded is used.

1: The maximum feedrate at which the allowable feedrate difference and acceleration for each axis are not exceeded is used. The feedrate is determined so that the decreased feedrate is constant regardless of the move direction when the profile is the same.

A feedrate is determined to satisfy the condition that the allowable feedrate difference and allowable acceleration rate of each axis are not exceeded, and also to ensure that a constant deceleration rate is applied to the same figure regardless of the direction of movement.

	#7	#6	#5	#4	#3	#2	#1	#0
19501			FRP					

[Input type] Parameter input
[Data type] Bit path

 # 5 FRP Linear rapid traverse is:
0: Acceleration/deceleration after interpolation
1: Acceleration/deceleration before interpolation
Set a maximum allowable acceleration rate for each axis in parameter No. 1671.
When using bell-shaped acceleration/deceleration before interpolation, set an acceleration rate change time in parameter No. 1672.

When this parameter is set to 1, acceleration/deceleration before interpolation is also applied to rapid traverse if all conditions below are satisfied. At this time, acceleration/deceleration after interpolation is not applied.
- Bit 1 (LRP) of parameter No. 1401 is set to 1: Linear interpolation type positioning
- A value other than 0 is set in parameter No. 1671 for an axis.
- The AI contour control mode is set.

If all of these conditions are not satisfied, acceleration/deceleration after interpolation is applied.

	#7	#6	#5	#4	#3	#2	#1	#0
19503				ZOL				HPF

[Input type] Parameter input
[Data type] Bit path

 # 0 HPF When a feedrate is determined based on acceleration in AI contour control, smooth feedrate control is:
0: Not used.
1: Used.

 # 4 ZOL The deceleration function based on cutting load in AI contour control (deceleration based on Z-axis fall angle) is:
0: Enabled for all commands.
1: Enabled for linear interpolation commands only.

	#7	#6	#5	#4	#3	#2	#1	#0
19515							ZG2	

[Input type] Parameter input
[Data type] Bit path

 # 1 ZG2 When the deceleration function based on cutting load in AI contour control (deceleration based on Z-axis fall angle) is used:
0: Stepwise override values are applied.
1: Inclined override values are applied.
This parameter is valid only when bit 4 (ZAG) of parameter No. 8451 is set to 1.
When this parameter is set to 1, be sure to set parameter Nos. 19516, 8456, 8457, and 8458.

19516	Override for area 1 in deceleration based on cutting load in AI contour control

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 100
This parameter sets an override value for area 1 when the deceleration function based on cutting load in AI contour control is used.
This parameter is valid only when bit 1 (ZG2) of parameter No. 19515 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
19530		CYS	CYA					

[Input type] Parameter input
[Data type] Bit path

 # 5 CYA Specifies whether to perform cylindrical interpolation cutting point compensation in the cylindrical interpolation command (G07.1).
0: Perform.
1: Do not perform.

 # 6 CYS Specifies whether when the cylindrical interpolation cutting point compensation function is used, cutting point compensation is performed between blocks or together with a block movement if the cutting point compensation value is less than the setting of parameter No. 19534.
0: Performed between blocks.
1: Performed together with a block movement if the cutting point compensation value is less than the setting of parameter No. 19534.

19534	Limit for changing cylindrical interpolation cutting point compensation in a single block

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 1 to 999999999

The following operation is performed, depending on the setting of parameter No.19530:

1) Parameter CYS (bit 6 of No. 19530) is set to 0
   If the amount of cylindrical interpolation cutting point compensation is smaller than the value set in this parameter, cylindrical interpolation cutting point compensation is not performed. Instead, this ignored amount of cylindrical interpolation cutting point compensation is added to the next amount of cylindrical interpolation cutting point compensation to determine whether to perform cylindrical interpolation cutting point compensation.

2) Parameter CYS (bit 6 of No. 19530) is set to 1
   If the amount of cylindrical interpolation cutting point compensation is smaller than the value set in this parameter, cylindrical interpolation cutting point compensation is performed together with the movement of the specified block.

NOTE
Set this parameter as follows:
Setting > (setting for a rotation axis in parameter No. 1422) × 4/3 where 4/3 is a constant for internal processing.

19535	Limit of travel distance moved with the cylindrical interpolation cutting point compensation in the previous block unchanged.

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 1 to 999999999

The following operation is performed, depending on the type of interpolation:

1) For linear interpolation
If the travel distance in a specified block is smaller than the value set in this parameter, machining is performed without changing the cylindrical interpolation cutting point compensation in the previous block.
2) For circular interpolation
If the diameter of a specified arc is smaller than the value set in this parameter, machining is performed without changing the cylindrical interpolation cutting point compensation in the previous block. Cylindrical interpolation cutting point compensation is not performed according to a circular movement.

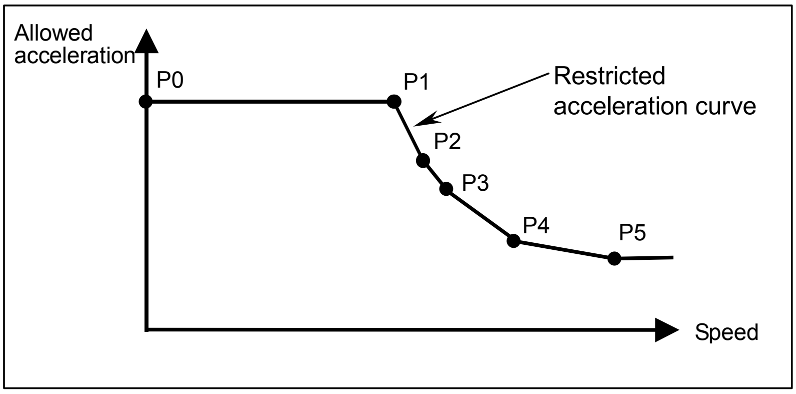
	#7	#6	#5	#4	#3	#2	#1	#0
19540								FAP

[Input type] Parameter input
[Data type] Bit path

 # 0 FAP Optimal torque acceleration/deceleration is:
0: Disabled.
1: Enabled.

When the linear positioning parameters, namely bit 1 (LRP) of parameter No. 1401 and bit 0 (FAP) of parameter No. 19540, are set to 1, and a value other than 0 is set in reference acceleration parameter (No. 1671) for an axis, the acceleration/deceleration for rapid traverse becomes optimal torque acceleration/deceleration in the mode for acceleration/deceleration before look-ahead interpolation (or the AI contour control mode). Optimal torque acceleration/ deceleration is controlled according to parameter-set restricted acceleration curve data.

Setting of restricted acceleration curve data



For each travel direction and each acceleration/deceleration operation, set the speed and allowable acceleration rate at each of the acceleration setting points (P0 to P5) for each axis in parameters. Set speeds in the speed parameters (No. 19541 to No. 19543). Set allowable acceleration rates in the allowable acceleration parameters (No. 19545 to No. 19568).

19541	**Optimal torque acceleration/deceleration (speed at P1)**
19542	**Optimal torque acceleration/deceleration (speed at P2)**
19543	**Optimal torque acceleration/deceleration (speed at P3)**
19544	**Optimal torque acceleration/deceleration (speed at P4)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01%
[Valid data range] 0 to 10000

The speeds at acceleration setting points P1 to P4 are to be set with speed parameters Nos. 19541 to 19544 as ratios to the rapid traverse speed (parameter No. 1420). The speed at P0 is 0, and the speed at P5 is the rapid traverse rate specified with parameter (No. 1420). Any acceleration setting point for which the speed parameter (one of Nos. 19541 to 19544) is set to 0 will be skipped.

19545	Optimal torque acceleration/deceleration (acceleration at P0 during movement in + direction and acceleration)

19546	Optimal torque acceleration/deceleration (acceleration at P1 during movement in + direction and acceleration)

19547	Optimal torque acceleration/deceleration (acceleration at P2 during movement in + direction and acceleration)

19548	Optimal torque acceleration/deceleration (acceleration at P3 during movement in + direction and acceleration)

19549	Optimal torque acceleration/deceleration (acceleration at P4 during movement in + direction and acceleration)

19550	Optimal torque acceleration/deceleration (acceleration at P5 during movement in + direction and acceleration)

19551	Optimal torque acceleration/deceleration (acceleration at P0 during movement in - direction and acceleration)

19552	Optimal torque acceleration/deceleration (acceleration at P1 during movement in - direction and acceleration)

19553	Optimal torque acceleration/deceleration (acceleration at P2 during movement in - direction and acceleration)

19554	Optimal torque acceleration/deceleration (acceleration at P3 during movement in - direction and acceleration)

19555	Optimal torque acceleration/deceleration (acceleration at P4 during movement in - direction and acceleration)

19556	Optimal torque acceleration/deceleration (acceleration at P5 during movement in - direction and acceleration)

19557	Optimal torque acceleration/deceleration (acceleration at P0 during movement in + direction and deceleration)

19558	Optimal torque acceleration/deceleration (acceleration at P1 during movement in + direction and deceleration)

19559	Optimal torque acceleration/deceleration (acceleration at P2 during movement in + direction and deceleration)

19560	Optimal torque acceleration/deceleration (acceleration at P3 during movement in + direction and deceleration)

19561	Optimal torque acceleration/deceleration (acceleration at P4 during movement in + direction and deceleration)

19562	Optimal torque acceleration/deceleration (acceleration at P5 during movement in + direction and deceleration)

19563	Optimal torque acceleration/deceleration (acceleration at P0 during movement in - direction and deceleration)

19564	Optimal torque acceleration/deceleration (acceleration at P1 during movement in - direction and deceleration)

19565	Optimal torque acceleration/deceleration (acceleration at P2 during movement in - direction and deceleration)

19566	Optimal torque acceleration/deceleration (acceleration at P3 during movement in - direction and deceleration)

19567	Optimal torque acceleration/deceleration (acceleration at P3 during movement in - direction and deceleration)

19568	Optimal torque acceleration/deceleration (acceleration at P5 during movement in - direction and deceleration)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01%
[Valid data range] 0 to 32767

For each travel direction and each acceleration/deceleration operation, set the allowable acceleration rate at each of the acceleration setting points (P0 to P5). As an allowable acceleration rate, set a ratio to the value set in the reference acceleration parameter (No. 1671). When 0 is set, the specification of 100% is assumed.

19581	Tolerance smoothing for nano smoothing

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch, degree (input unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets a tolerance value for a program created using miniature line segments in nano smoothing.

When 0 is set in this parameter, a minimum amount of travel in the increment system is regarded as a tolerance value.

19582	Minimum amount of travel of a block that makes a decision based on an angular difference between blocks for nano smoothing

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch, degree (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the minimum amount of travel of a block that makes a decision based on an angular difference between blocks for nano smoothing. A block that specifies an amount of travel less than the value set in this parameter makes no decision based on an angular difference.

When 0 is set in this parameter, a decision based on an angular difference is made with all blocks.

A value greater than the value set in parameter No. 8490 for making a decision based on the minimum travel distance of a block must be set.

	#7	#6	#5	#4	#3	#2	#1	#0
19607			CAV				SPG	
			CAV				SPG	

[Input type] Parameter input
[Data type] Bit path

 # 1 SPG To apply cutter compensation for 5-axis machining to a machine having a table rotation axis, as the G code to be specified:
0: G41.2/G42.2 is used regardless of the machine type.
1: G41.4/G42.4 is used for a table rotation type machine; G41.5/G42.5 for a mixed type machine.

 # 5 CAV When an interference check finds that interference (overcutting) occurred:
0: Machining stops with the alarm (PS0041).
(Interference check alarm function)
1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)

For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

	#7	#6	#5	#4	#3	#2	#1	#0
19608		MIR	PRI			DET	NI5	

[Input type] Parameter input
[Data type] Bit path

#1 NI5 For an interference check of cutter compensation for 5-axis machining:

0: The specified position in the workpiece coordinate system and compensation vector are used.
The interference check avoidance function cannot be used.

1: The position at which the programmed command specified with the table coordinate system is focused onto the plane normal to the tool axis direction and the compensation vector are used.
The interference check avoidance function can be used.

 # 2 DET When the programming coordinate system is fastened to the table in tool tip point control for 5-axis machining or cutter compensation for 5-axis, the relative position and absolute position of a specified path are:

0: Displayed in the programming coordinate system (fastened to the table).

1: Displayed in the workpiece coordinate system (not fastened to the table).

 # 5 PRI Among multiple end point candidates that exist when a movement is made on a rotation axis by a command such as I, J, and K when a slanted surface machining command is specified under tool tip point control for 5-axis machining (type 2) or cutter compensation for 5-axis (type 2):

0: A combination in which the master (first rotation axis) makes a smaller angular movement is selected for a machine of tool rotation type or table rotation type. A combination in which the table (second rotation axis) makes a smaller angular movement is selected for a machine of composite type.

1: A combination in which the slave (second rotation axis) makes a smaller angular movement is selected for a machine of tool rotation type or table rotation type. A combination in which the tool (first rotation axis) makes a smaller angular movement is selected for a machine of composite type.

 # 6 MIR When programmable mirror image is applied to a linear axis in tool tip point control for 5-axis machining (type 2) or cutter compensation for 5-axis (type 2), mirror image is:

0: Not applied to a specified I, J, or K command

1: Applied to a specified I, J, or K command.

19631	Variation in determining an angle for leading edge offset

[Input type] Parameter input
[Data type] Real path
[Unit of data] degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set a variation range for determining the included angle between the tool vector (VT) and movement vector (VM) to be 0°, 180°, or 90° in the leading edge offset function.

For example, let the included angle between VT and VM be θ (0≤θ≤180) and the angle set in this parameter be Δθ. Then, θ is determined as follows:

If 0≤θ≤Δθ: θ=0°
If (180−Δθ)≤θ≤180: θ=180°
If (90−Δθ)≤θ≤(90+Δθ):θ=90°

Normally, a value around 1.0 is set.

19632	Distance from a programmed point (pivot point) to tool tip position (cutting point)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the distance from a programmed point to actual cutting point to allow vector calculation for cutter compensation for 5-axis machining at the tool tip.

If this parameter is set to 0, the cutter compensation function for 5-axis machining cannot be performed at the tool tip.

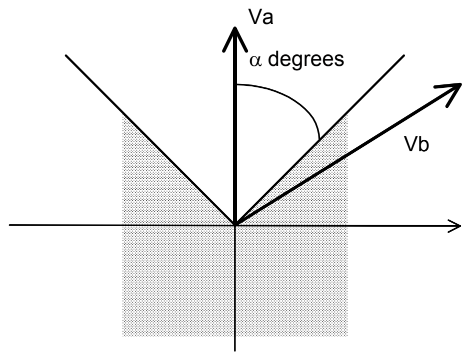
NOTE
When changing the setting of this parameter, make the change before turning on the cutter compensation mode for 5-axis machining.

19635	Angle for determination in interference checks in cutter compensation for 5-axis machining

[Input type] Parameter input
[Data type] Real path
[Unit of data] degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

In cutter compensation for 5-axis machining, if the difference in angle between two tool vectors is greater than or equal to the setting in this parameter, the tool direction is regarded as having changed.
If 0 is set, 45 degrees is assumed.

Let two tool vectors be Va and Vb. If the difference in angle is α degrees or greater as shown in the figure below, the tool vector is regarded as having changed.



19636	Angle used to determine whether to execute the interference check/avoidance function of cutter compensation for 5-axis machining

[Input type] Setting input
[Data type] Real path
[Unit of data] deg
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))

The interference check/avoidance function of cutter compensation for 5-axis machining is executed when the angle difference between the tool direction vectors for the target two points is less than the setting.

This parameter is valid when bit 1 (NI5) of parameter No. 19608 is set to 1. When the setting is 0, the angle is assumed to be 10.0 degrees.

19658	Angular displacement of a rotation axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the coordinate of a rotation axis, among the rotation axes determining the tool axis direction, which is not controlled by the CNC for the tool axis direction tool length compensation function. Whether this parameter is valid or invalid is determined by the setting of bit 1 (RAP) of parameter No. 19650.

19659	Offset value for the angular displacement of a rotation axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
An offset can be applied to the angular displacement for the tool axis direction tool length compensation function to compensate for the move direction.

19660	Origin offset value of a rotation axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets an angular displacement shifted from the origin for a rotation axis for the tool axis direction tool length compensation function.

19661	Rotation center compensation vector in tool axis direction tool length compensation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the vector from the first rotation axis center to the second rotation axis center for the tool axis direction tool length compensation function.

19662	Spindle center compensation vector in tool axis direction tool length compensation

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

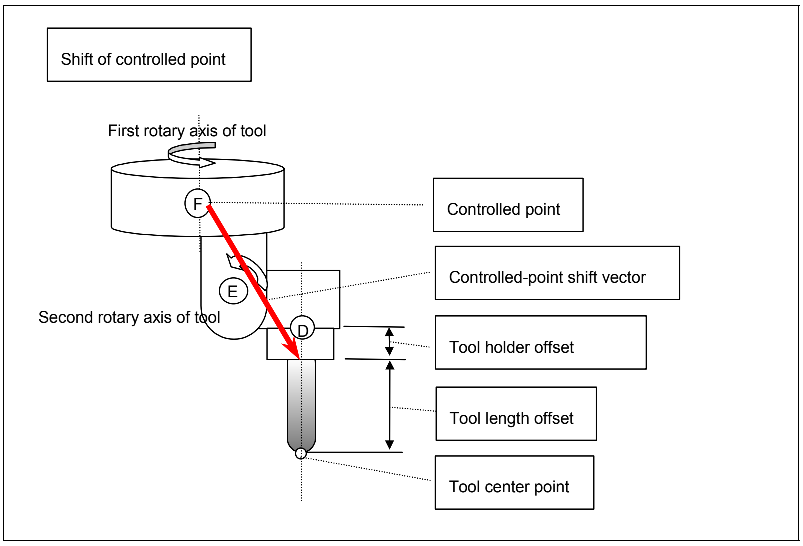
This parameter sets the compensation vector for the spindle center for the tool axis direction tool length compensation function.

	#7	#6	#5	#4	#3	#2	#1	#0
19665			SVC	SPR				

[Input type] Parameter input
[Data type] Bit path

 # 4 SPR The controlled point is shifted by:
0: Automatic calculation.
1: Using parameter No. 19667.

SVC (bit 5 of parameter No. 19665)	SPR (bit 4 of parameter No. 19665)	Shift of controlled point
0	-	Shift is not performed as not done conventionally.
1	0	The controlled point is shifted according to the result of the following automatic calculation: - (Intersection offset vector between the tool axis and the first rotation axis of the tool + intersection offset vector between the second and first rotation axes of the tool + tool holder offset (parameter No. 19666)) (See the figure below.)
1	1	The controlled point is shifted. As the shift vector, the vector set in parameter No. 19667 is used.



[Controlled-point shift vector when automatically calculated]

 # 5 **SVC** The controlled point is:

0: Not shifted.

1: Shifted.

The method of shifting is specified by bit 4 (SPR) of parameter No. 19665.

NOTE

When the machine has no rotation axis for rotating the tool (when parameter No. 19680 is set to 12 to specify the table rotation type), the controlled point is not shifted regardless of the setting of this parameter.

19666	Tool holder offset value

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

When the tool axis direction tool length compensation function, tool tip center rotation handle feed/interrupt, and the display of the tool center point position are performed, specify the offset for the machine-specific section from the rotation center of the rotation axis to the tool mounting position (the tool holder offset value) in tool length compensation during tool center point control, tool center point control for 5-axis machining, and tilted working plane command mode (after G53.1). For the tool axis direction tool length compensation function, the tool holder offset function can be enabled or disabled by setting bit 7 (ETH) of parameter No. 19665.

NOTE
Set a radius value.

19667	Controlled-point shift vector

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the shift vector for the controlled point. This value becomes valid when bit 5 (SVC) of parameter No. 19665 is set to 1, and bit 4 (SPR) of parameter No. 19665 is set to 1.

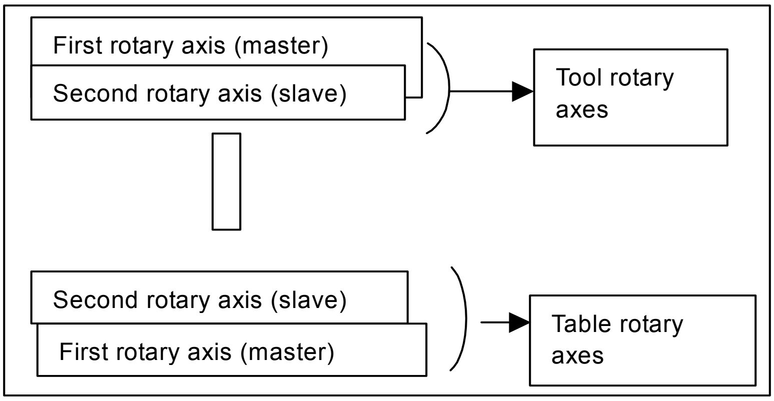
NOTE
Set a radius value.

19680	Mechanical unit type

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 21

Specify the type of the mechanical unit.

Parameter No. 19680	Mechanical unit type	Controlled rotation axis	Master and slave
0		Mechanism having no rotation axis	
2	Tool rotation type	Two rotation axes of the tool	The first rotation axis is the master, and the second rotation axis is the slave.
12	Table rotation type	Two rotation axes of the table	The first rotation axis is the master, and the second rotation axis is the slave.
21	Mixed type	One rotation axis of the tool + one rotation axis of the table	The first rotation axis is the tool rotation axis, and the second rotation axis is the table rotation axis.



NOTE

A hypothetical axis is also counted as a controlled rotary axis.

<Hypothetical axis>

In some cases, it is convenient to use an imaginary rotary axis whose angle is fixed to a certain value. For example, suppose that a tool is mounted in a tilted manner through an attachment. In such a case, the rotary axis considered hypothetically is a hypothetical axis. Bits 0 and 1 of parameter No. 19696 determine whether each rotary axis is an ordinary roatry axis or a hypothetical axis.

19681	Controlled-axis number for the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the controlled-axis number for the first rotation axis.
For a hypothetical axis (when bit 0 (IA1) of parameter No. 19696 is 1), set 0.

19684	Rotation direction of the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1

Set the direction in which the first rotation axis rotates as a mechanical motion when a positive move command is issued.

0: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (right-hand thread rotation)
1: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (left-hand thread rotation)

Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19686	Controlled-axis number for the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the controlled-axis number for the second rotation axis.
For a hypothetical axis (bit 1 (IA2) of parameter No. 19696 is 1), set 0.

	#7	#6	#5	#4	#3	#2	#1	#0
19696		RFC	WKP					

[Input type] Parameter input
[Data type] Bit path

 # 5 WKP For a 5-axis machine having a table rotation axis, as the programming coordinate system for tool tip point control for 5-axis machining or cutter compensation for 5-axis machining:

0: The table coordinate system (coordinate system fixed on the rotary table) is used.
1: The workpiece coordinate system is used.

> **NOTE**
> For cutter compensation for 5-axis machining, the setting of this parameter is used only when bit 4 (TBP) of parameter No. 19746 is set to 1.

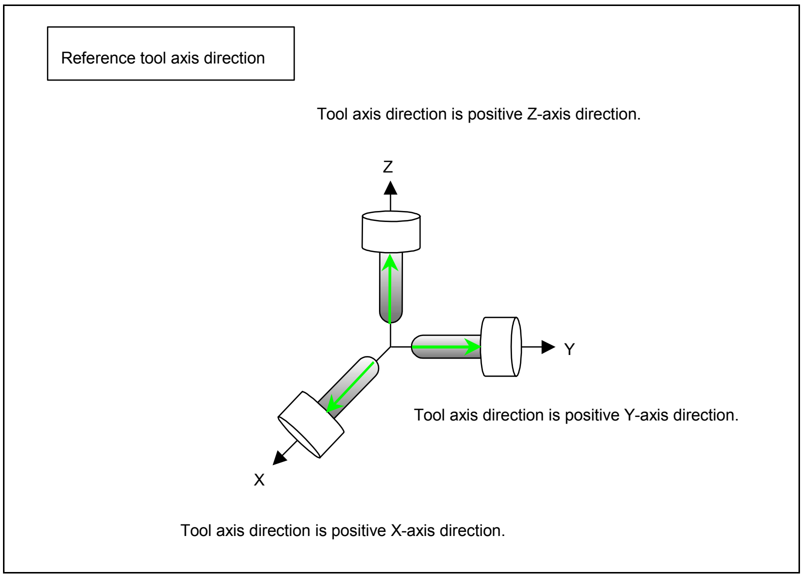
 # 6 **RFC** In tool center point control for 5-axis machining, when a command that does not move the tool center point with respect to the workpiece is issued, the feedrate of the rotation axis is:
0: The maximum cutting feedrate (parameter No. 1422).
1: A specified feedrate.

19697	Reference tool axis direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Set the tool axis direction in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees. Also, set the tool axis direction in the machine coordinate system in a mechanism in which only the rotation axes for controlling the table are present (there is no rotation axis for controlling the tool).
1: Positive X-axis direction
2: Positive Y-axis direction
3: Positive Z-axis direction

When the reference tool axis direction is neither the X-, Y-, nor Z-axis direction, set the reference direction in this parameter, then set appropriate angles as the reference angle RA and reference angle RB (parameter Nos. 19698 and 19699).



19698	Angle when the reference tool axis direction is tilted (reference angle RA)

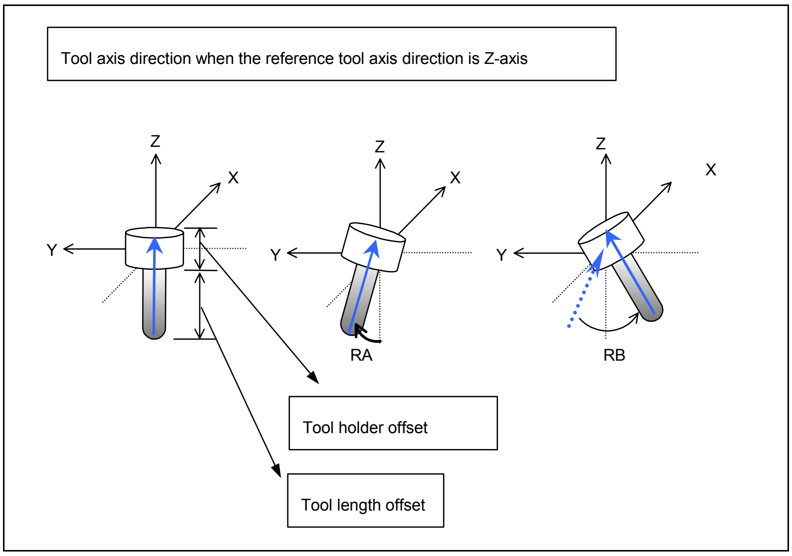
19699	Angle when the reference tool axis direction is tilted (reference angle RB)

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)

When the reference tool axis direction (parameter No. 19697) is set to 1, the tool axis is tilted the RA degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction, then the tool axis is tilted the RB degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction.

When the reference tool axis direction (parameter No. 19697) is set to 2, the tool axis is tilted the RA degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction, then the tool axis is tilted the RB degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction.

When the reference tool axis direction (parameter No. 19697) is set to 3, the tool axis is tilted the RA degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction, then the tool axis is tilted the RB degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction.



19712	**Intersection offset vector between the second and first rotation axes of the tool (X-axis of the basic three axes)**

19713	**Intersection offset vector between the second and first rotation axes of the tool (Y-axis of the basic three axes)**

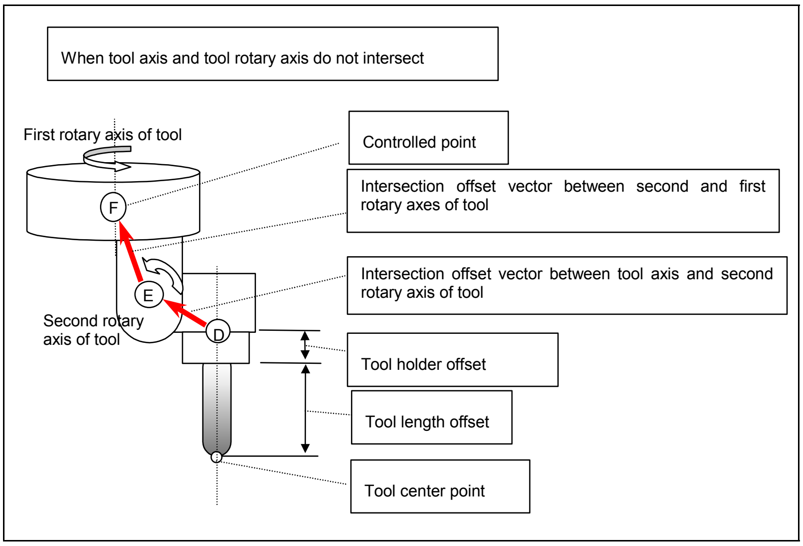
19714	**Intersection offset vector between the second and first rotation axes of the tool (Z-axis of the basic three axes)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set these parameters when the rotation axes of the tool do not intersect.

These parameters are valid when parameter No. 19680 is set to 2.

Set the vector from point E on the second rotation axis of the tool to point F on the first rotation axis of the tool as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.



NOTE

As point F, set a position that is easy to measure.
Set a radius value.

19741	Upper limit of the movement range of the first rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)

In tool center point control for 5-axis machining (type 2), cutter compensation for 5-axis (type 2), or tilted working plane command, set the upper limit of the movement range of the first rotation axis. When the movement range of the first rotation axis is not specified, this parameter and parameter No. 19742 must both be set to 0.

19742	Lower limit of the movement range of the first rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)

In tool center point control for 5-axis machining (type 2), cutter compensation for 5-axis (type 2), or tilted working plane command, set the lower limit of the movement range of the first rotation axis. When the movement range of the first rotation axis is not specified, this parameter and parameter No. 19741 must both be set to 0.

19743	Upper limit of the movement range of the second rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)

In tool center point control for 5-axis machining (type 2), cutter compensation for 5-axis (type 2), or tilted working plane command, set the upper limit of the movement range of the second rotation axis. When the movement range of the second rotation axis is not specified, this parameter and parameter No. 19744 must both be set to 0.

19744	Lower limit of the movement range of the second rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)

In tool center point control for 5-axis machining (type 2), cutter compensation for 5-axis (type 2), or tilted working plane command, set the lower limit of the movement range of the second rotation axis. When the movement range of the second rotation axis is not specified, this parameter and parameter No. 19743 must both be set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
19746		CRS		TBP	LOZ	LOD	PTD	

[Input type] Parameter input
[Data type] Bit path

 # 1 PTD When cutter compensation for 5-axis machining is performed for a machine of table rotation type, the tool direction is specified:
0: Using parameters No. 19697, No. 19698, and No. 19699.
1: As a direction perpendicular to the plane specified by G17/G18/G19.

 # 2 LOD As the tool length for 5-axis machining manual feed:
0: The value of parameter No. 12318 is used.
1: The tool length currently used for tool length compensation is used.

 # 3 LOZ When bit 2 (LOD) of parameter No. 19746 is set to 1 and tool length compensation is not applied, as the tool length for 5-axis machining manual feed:
0: The value of parameter No. 12318 is used.
1: 0 is used.

 # 4 TBP For a 5-axis machine having a table rotation axis, as the programming coordinate system for cutter compensation for 5-axis machining:
0: The workpiece coordinate system is used.
1: The setting of bit 5 (WKP) of parameter No. 19696 is used.

 # 6 CRS In tool tip point control for 5-axis machining, when the deviation from the path during movement at the specified cutting feedrate or rapid traverse rate is determined to exceed the limit:

0: The feedrate or rapid traverse rate is not decreased.
1: The feedrate or rapid traverse rate is controlled so that the limit of the deviation from the path set in the parameter for the cutting feed or rapid traverse is not exceeded.

When this parameter is set to 1:
In the rapid traverse mode, the rapid traverse rate is decreased so that the deviation from the path does not exceed the limit specified in parameter No. 19751.
In the cutting feed mode, the cutting feedrate is decreased so that the deviation from the path does not exceed the limit specified in parameter No. 19752.

19751	Limit of the deviation from the path (for rapid traverse)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the limit of the deviation from the path in the rapid traverse mode in tool tip point control for 5-axis machining.
If the tool moves at the specified rate, the deviation from the path may exceed the value specified in this parameter. In this case, the rate is decreased so that the tool moves along the path.
This parameter is valid when bit 6 (CRS) of parameter No. 19746 is set to 1.
When 0 is set, the least input increment is assumed to be the limit of the deviation from the path.
If a negative value is set, the rapid traverse rate is not decreased.

NOTE
The error generated after the rate is decreased may be smaller than the value set in this parameter depending on the calculation error.

19752	Limit of the deviation from the path (for cutting feed)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the limit of the deviation from the path in the cutting feed mode in tool tip point control for 5-axis machining.

If the tool moves at the specified rate, the deviation from the path may exceed the value specified in this parameter. In this case, the rate is decreased so that the tool moves along the path.

This parameter is valid when bit 6 (CRS) of parameter No. 19746 is set to 1.

When 0 is set, the least input increment is assumed to be the limit of the deviation from the path.

If a negative value is set, the cutting feedrate is not decreased.

NOTE

The error generated after the rate is decreased may be smaller than the value set in this parameter depending on the calculation error.

A.2 DATA TYPE

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks
Bit Bit machine group Bit path Bit axis Bit spindle	0 or 1	
Byte Byte machine group Byte path Byte axis Byte spindle	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Word Word machine group Word path Word axis Word spindle	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
2-word 2-word machine group 2-word path 2-word axis 2-word spindle	0 to ±999999999	Some parameters handle these types of data as unsigned data.
Real Real machine group Real path Real axis Real spindle	See the standard parameter setting tables.	

NOTE

1. Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
2. For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group.
3. For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
4. For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
5. For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
6. The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

A.3 STANDARD PARAMETER SETTING TABLES

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

NOTE
1. Values are rounded up or down to the nearest multiples of the minimum data unit.
2. A valid data range means data input limits, and may differ from values representing actual performance.
3. For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value."

(A) Length and angle parameters (type 1)

Unit of data	Increment	Minimum	Valid data range
mm deg.	IS-A	0.01	-999999.99 to +999999.99
	IS-B	0.001	-999999.999 to +999999.999
	IS-C	0.0001	-99999.9999 to +99999.9999
	IS-D	0.00001	-9999.99999 to +9999.99999
	IS-E	0.000001	-999.999999 to +999.999999
inch	IS-A	0.001	-99999.999 to +99999.999
	IS-B	0.0001	-99999.9999 to +99999.9999
	IS-C	0.00001	-9999.99999 to +9999.99999
	IS-D	0.000001	-999.999999 to +999.999999
	IS-E	0.0000001	-99.9999999 to +99.9999999

(B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
inch	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999
	IS-D	0.000001	0.000000 to +999.999999
	IS-E	0.0000001	0.0000000 to +99.9999999

(C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-A	0.01	0.00 to +999000.00
	IS-B	0.001	0.000 to +999000.000
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
inch/min	IS-A	0.001	0.000 to +96000.000
	IS-B	0.0001	0.0000 to +9600.0000
	IS-C	0.00001	0.00000 to +4000.00000
	IS-D	0.000001	0.000000 to +400.000000
	IS-E	0.0000001	0.0000000 to +40.0000000

(D)Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 $deg./sec^2$	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
$inch/sec^2$	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999
	IS-D	0.000001	0.000000 to +999.999999
	IS-E	0.0000001	0.0000000 to +99.9999999

B PROGRAM CODE LIST

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	without custom macro	with custom macro	
Number 0	0	30	0	20			*
Number 1	1	B1	1	01			*
Number 2	2	B2	2	02			*
Number 3	3	33	3	13			*
Number 4	4	B4	4	04			*
Number 5	5	35	5	15			*
Number 6	6	36	6	16			*
Number 7	7	B7	7	07			*
Number 8	8	B8	8	08			*
Number 9	9	39	9	19			*
Address A	A	41	a	61			*
Address B	B	42	b	62			*
Address C	C	C3	c	73			*
Address D	D	44	d	64			*
Address E	E	C5	e	75			*
Address F	F	C6	f	76			*
Address G	G	47	g	67			*
Address H	H	48	h	68			*
Address I	I	C9	i	79			*
Address J	J	CA	j	51			*
Address K	K	4B	k	52			*
Address L	L	CC	l	43			*
Address M	M	4D	m	54			*
Address N	N	4E	n	45			*
Address O	O	CF	o	46			*
Address P	P	50	p	57			*
Address Q	Q	D1	q	58			*
Address R	R	D2	r	49			*
Address S	S	53	s	32			*
Address T	T	D4	t	23			*
Address U	U	55	u	34			*
Address V	V	56	v	25			*
Address W	W	D7	w	26			*
Address X	X	D8	x	37			*
Address Y	Y	59	y	38			*
Address Z	Z	5A	z	29			*
Delete	DEL	FF	Del	7F	×	×	
Back space	BS	88	BS	2A	×	×	
Tabulator	HT	09	Tab	2E	×	×	
End of block	LF or NL	0A	CR or EOB	80			
Carriage return	CR	8D			×	×	

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	without custom macro	with custom macro	
Space	SP	A0	SP	10	□	□	
Absolute rewind stop	%	A5	ER	0B			
Control out (start of comment)	(	28	(2-4-5)	1A			
Control in (end of comment)	)	A9	(2-4-7)	4A			
Plus sign	+	2B	+	70	△		*
Minus sign	-	2D	-	40			*
Colon (address O)	:	3A					
Optional block skip	/	AF	/	31			
Period (decimal point)	.	2E	.	6B			*
Sharp	#	A3	Parameter (No.6012)				
Dollar sign	$	24			×	×	
Ampersand	&	A6	&	0E	△	○	
Apostrophe	’	27			△	△	
Asterisk	*	AA	Parameter (No.6010)		△		
Comma	,	AC	,	3B			
Semicolon	;	FB			×	×	
Left angle bracket	<	2C					
Equal sign	=	BD	Parameter (No.6011)		△		
Right angle bracket	>	BE					
Question mark	?	3F			△	○	
Commercial at mark	@	C0			△	○	
Quotation mark	“	22			△	△	
Left square bracket	[	DB	Parameter (No.6013)		△		
Right square bracket	]	DD	Parameter (No.6014)		△		
Underscore	_	6F	Parameter (No.6018)				*
Lowercase letter a	a	E1			△	△	*
Lowercase letter b	b	E2			△	△	*
Lowercase letter c	c	63			△	△	*
Lowercase letter d	d	E4			△	△	*
Lowercase letter e	e	65			△	△	*
Lowercase letter f	f	66			△	△	*
Lowercase letter g	g	E7			△	△	*
Lowercase letter h	h	E8			△	△	*
Lowercase letter i	i	69			△	△	*
Lowercase letter j	j	6A			△	△	*
Lowercase letter k	k	EB			△	△	*
Lowercase letter l	l	6C			△	△	*
Lowercase letter m	m	ED			△	△	*
Lowercase letter n	n	EE			△	△	*
Lowercase letter o	o	6F			△	△	*

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	without custom macro	with custom macro	
Lowercase letter p	p	F0			△	△	*
Lowercase letter q	q	71			△	△	*
Lowercase letter r	r	72			△	△	*
Lowercase letter s	s	F3			△	△	*
Lowercase letter t	t	74			△	△	*
Lowercase letter u	u	F5			△	△	*
Lowercase letter v	v	F6			△	△	*
Lowercase letter w	w	77			△	△	*
Lowercase letter x	x	78			△	△	*
Lowercase letter y	y	F9			△	△	*
Lowercase letter z	z	FA			△	△	*

NOTE

1 The symbols used in the "Custom macro B" column have the following meanings.
   (Space) : The character will be registered in memory and has a specific meaning. It is used incorrectly in a statement other than a comment, an alarm occurs.
   × : The character will not be registered in memory and will be ignored.
   △ : The character will be registered in memory, but will be ignored during program execution. However, a character indicated with "*" will not be ignored when used in a file name.
   ○ : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.
   □ : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.

2 The symbol used in the column "Usable as file name" has the following meaning:
   *: Capable of being coded between "<" and ">" as a file name.

3 Codes not in this table are ignored if their parity is correct.

4 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

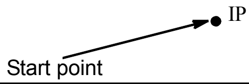
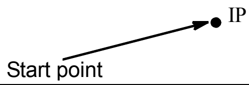
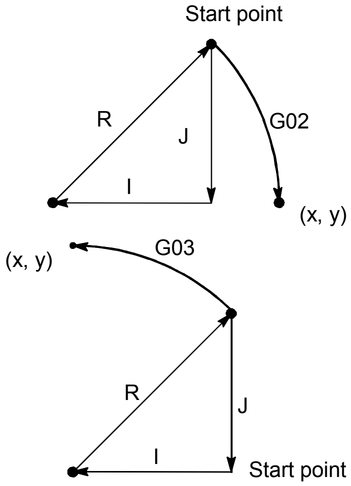
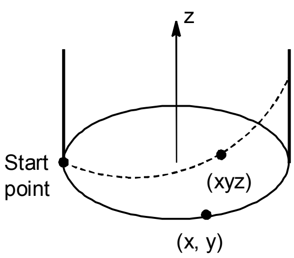
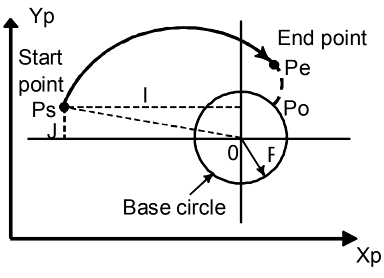
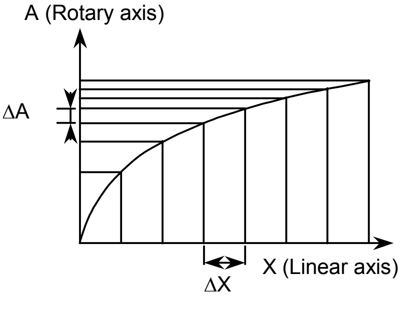
C LIST OF FUNCTIONS AND PROGRAM FORMAT

With some functions, the format used for specification on the machining center system differs from the format used for specification on the lathe system. Moreover, some functions are used for only one of the control types for the machining center system and lathe system.
Some functions cannot be added as options depending on the model.
For details of command formats, see the relevant sections or subsections.

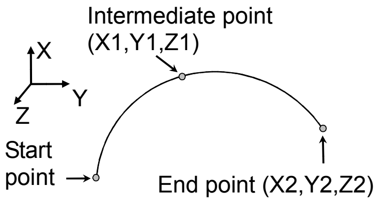
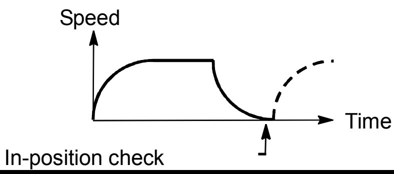
In the list, the following symbols are used:

- For machining center
  x : 1st basic axis (X),
  y : 2nd basic axis (Y),
  z : 3rd basic axis (Z)
- For lathe
  x : 1st basic axis (X),
  z : 2nd basic axis (Z),
  coded using G code system A
- IP_ : presents a combination of arbitrary axis addresses using X,Y,Z,A,B and C (such as X_ Y_ Z_).

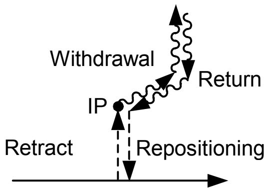
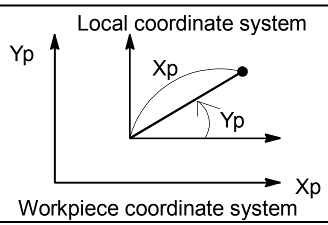
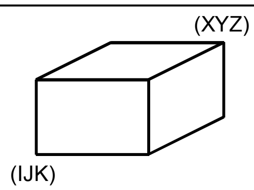
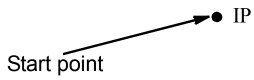
(1/10)

Functions	Illustration	Program format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		• **For machining center** G17 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Y_ $\begin{Bmatrix} R\_ \\ I\_\ J\_ \end{Bmatrix}$ F_ ; G18 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Z_ $\begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix}$ F_ ; G19 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ Y_ Z_ $\begin{Bmatrix} R\_ \\ J\_\ K\_ \end{Bmatrix}$ F_ ; • **For lathe** $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Z_ $\begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix}$ F_ ;
Helical interpolation (G02, G03)	 In case of G03 on X-Y plane	G17 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Y_ $\begin{Bmatrix} R\_ \\ I\_\ J\_ \end{Bmatrix}$ α_ F_ ; G18 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Z_ $\begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix}$ α_ F_ ; G19 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ Y_ Z_ $\begin{Bmatrix} R\_ \\ J\_\ K\_ \end{Bmatrix}$ α_ F_ ; α : Any axis other than circular interpolation axes.
Involute interpolation (G02.2, G03.2)	 (In case of X-Y plane)	G17 $\begin{Bmatrix} G02.2 \\ G03.2 \end{Bmatrix}$ Xp_ Yp_ I_ J_ R_ F_ ; G18 $\begin{Bmatrix} G02.2 \\ G03.2 \end{Bmatrix}$ Zp_Xp_ K_ I_ R_ F_ ; G19 $\begin{Bmatrix} G02.2 \\ G03.2 \end{Bmatrix}$ Yp_ Zp_ J_ K_ R_ F_ ;
Exponential interpolation (G02.3, G03.3)	 Relationship between X-axis and A-axis	Positive rotation G02.3 X_ Y_ Z_ I_ J_ K_ R_ F_ Q_; Negative rotation G03.3 X_ Y_ Z_ I_ J_ K_ R_ F_ Q_;

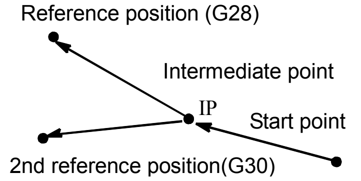
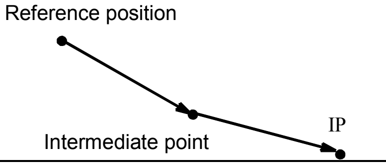
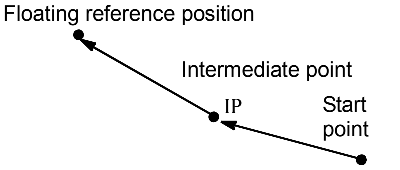
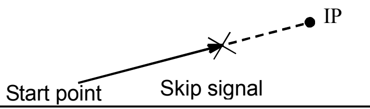
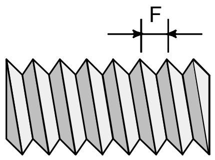
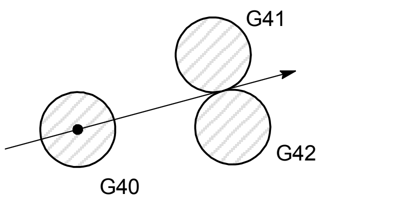
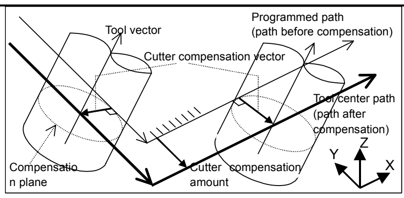
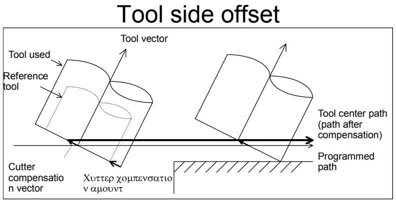
(2/10)

Functions	Illustration	Program format
Three-dimensional circular interpolation (G02.4, G03.4)		G02.4 $X_{X1}\ Y_{Y1}\ Z_{Z1}\ \alpha_{\alpha1}\ \beta_{\beta1}$; First block (mid-point of the arc) $X_{X2}\ Y_{Y2}\ Z_{Z2}\ \alpha_{\alpha2}\ \beta_{\beta2}$; Second block (end point of the arc) α, β : Arbitrary axes other than the 3-dimensional circular interpolation axis (up to two axes) G03.4 can also be specified instead of G02.4.
Dwell (G04)		G04 $\left\{\begin{array}{l} X\_ \\ P\_ \end{array}\right\}$;
AI contour control (G05)		G05 P10000 ; AI contour control start G05 P0 ; AI contour control end
AI contour control (G05.1)		G05.1 Q1 ; AI contour control mode on G05.1 Q0 ; AI contour control mode off
Nano smoothing (G05.1)		G05.1 Q3 IP0 ; Nano smoothing mode on G05.1 Q0 ; Nano smoothing mode off
Smooth interpolation (G05.1)		G05.1 Q2 ; Smooth interpolation mode on G05.1 Q0 ; Smooth interpolation mode off
NURBS interpolation (G06.2)		G06.2[P_] K_ IP_ [R_] [F_]; NURBS interpolation mode on P : Rank of NURBS curve IP : Control point R : Weight K : Knot F : Feedrate
Hypothetical axis interpolation (G07)		G07 IP0 ; Hypothetical axis setting G07 IP1 ; Hypothetical axis cancel
Cylindrical interpolation (G07.1)		G07 IP_ r_; Cylindrical interpolation mode r : Cylinder radius G07 IP 0; Cylindrical interpolation mode cancel
AI contour control (Advanced preview control) (G08)		G08 P1 ; AI contour control mode on G08 P0 ; AI contour control mode off
Exact stop (G09)		G09 $\left\{\begin{array}{c} G01 \\ G02 \\ G03 \end{array}\right\}$ IP_ ;

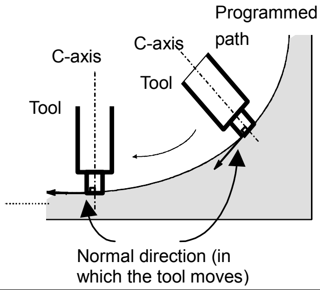
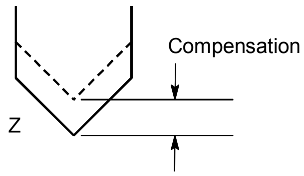
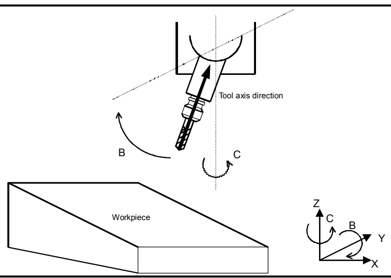
(3/10)

Functions	Illustration	Program format
Programmable data input (G10)		**• For machining center** Tool compensation memory A G10 L01 P_ R_ ; Tool compensation memory B G10 L10 P_ R_ ; (Geometry offset amount) G10 L11 P_ R_ ; (Wear offset amount) Tool compensation memory C G10 L10 P_ R_ ; (Geometry offset amount/H) G10 L11 P_ R_ ; (Wear offset amount/H) G10 L12 P_ R_ ; (Geometry offset amount/D) G10 L13 P_ R_ ; (Wear offset amount/D) **• For lathe** Geometry offset amount G10 P_ X_ Z_ R_ Q_ ; P = 10000 + Geometry offset number Wear offset amount G10 P_ X_ Z_ C_ Q_ ; P = Wear offset number
Tool retract and recover (G10.6)		G10.6 IP_ ; Specify the amount of retraction G10.6 (as a single block containing no other commands) ; Cancel the amount of retraction
Polar coordinate interpolation (G12.1, G13.1)		G12.1 ; Polar coordinate interpolation mode on G13.1 ; Polar coordinate interpolation cancel
Polar coordinate command (G15, G16)		G17 G16 Xp_ Yp_ . . . ; G18 G16 Zp_ Xp_ . . . ; G19 G16 Yp_ Zp_ . . . ; G15 ; Cancel
Plane selection (G17, G18, G19)		G17 ; Xp Yp-plane selection G18 ; Zp Xp-plane selection G19 ; Yp Zp-plane selection
Inch/metric conversion (G20, G21)		Inch input G20 ; Metric input G21 ;
Stored stroke check (G22, 23)		G22 X_ Y_ Z_ I_ J_ K_ ; G23 ; Cancel
Reference position return check (G27)		G27 IP_ ;

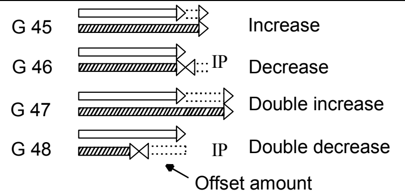
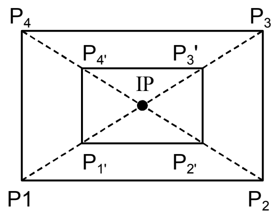
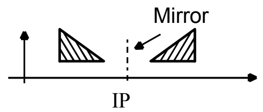
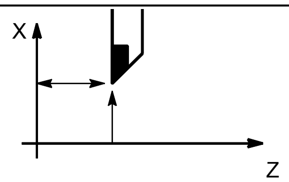
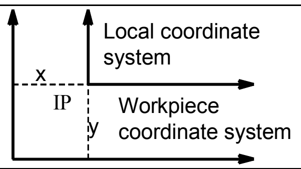
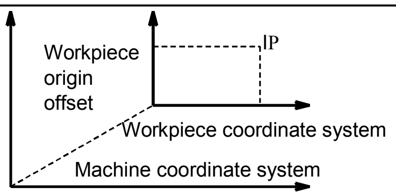
(4/10)

Functions	Illustration	Program format
Reference position return (G28) 2nd Reference position return (G30)		G28 IP_ ; G30 IP_ ;
Movement from reference position (G29)		G29 IP_ ;
Floating reference position return (G30.1)		G30.1 IP_ ;
Skip function (G31)		G31 IP_ F_ ;
Threading (G33) Threading (G32)		• **For machining center** G33 IP_ F_ ; F : Lead • **For lathe** Equal lead threading G32 P_ F_ ;
Cutter or tool nose radius compensation, three-dimensional cutter compensation (G38, G39, G40 to G42)		• **For machining center** $\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}$ $\left\{\begin{matrix}G41\\G42\end{matrix}\right\}$ D_ ; D : Tool compensation number G40 : Cancel
Cutter or tool nose radius compensation (G40 to G42) (G38, G39)		• **For lathe only** $\left\{\begin{matrix}G41\\G42\end{matrix}\right\}$ IP_ ; G40 : Cancel
Cutter compensation for 5-axis machining (G41.2, G42.2) (G41.3)	 Tool side offset  Leading edge offset	$\left\{\begin{matrix}G41.2\\G42.2\end{matrix}\right\}$ IP_ D_ ; Tool side offset G40 IP_ ; Cancellation of cutter compensation G41.3 D_ ; Leading edge offset G40 ; Canceling the leading edge offset

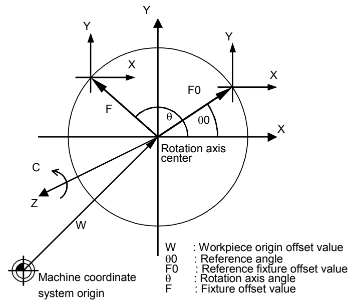
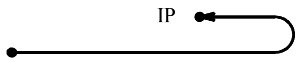
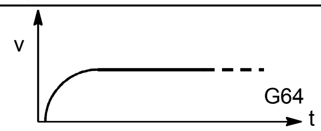
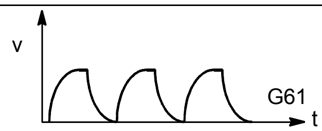
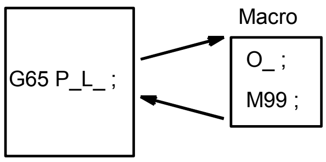
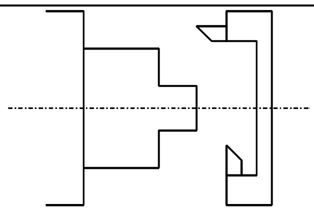
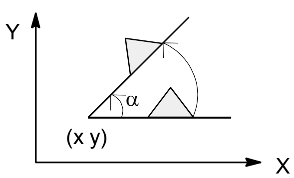
(5/10)

Functions	Illustration	Program format
Normal direction control (G40.1, G41.1, G42.1)		G41.1 ; Normal direction control on : right G42.1 ; Normal direction control on : left G40.1 ; Normal direction control cancel
Tool length compensation (G43, G44, G49)		$\left\{\begin{array}{l}G43\\G44\end{array}\right\}$ Z_ H_ ; $\left\{\begin{array}{l}G43\\G44\end{array}\right\}$ H_ ; H : Tool compensation number G49 : Cancel
Tool length compensation in tool axis direction (G43.1)		G43.1 H_ ; Tool length compensation in tool axis direction H : Compensation number G49 ; Compensation cancel
Tool center point control (TYPE1) (G43.4)		G43.4 IP α β H ; Tool center point control (TYPE1) start IP α β ; IP : In the case of an absolute command, the coordinate value of the end point of the tool tip movement In the case of an incremental command, the amount of the tool tip movement α, β :In the case of an absolute command, the coordinate value of the end point of the rotation axes In the case of an incremental command, the amount of the rotation axis movement H : Tool compensation number
Tool center point control (TYPE2) (G43.5)		G43.5 IP_ H_ Q_ ; Tool center point control (TYPE2) start IP_ I_ J_ K_ ; IP : In the case of an absolute command, the coordinate value of the end point of the tool tip movement In the case of an incremental command, the amount of the tool tip movement I,J,K : Tool axis direction at the block end point as seen from the programming coordinate system H : Tool compensation number Q : Inclination angle of the tool (in degrees)
Tool offset (G43.7)		• **For lathe only**

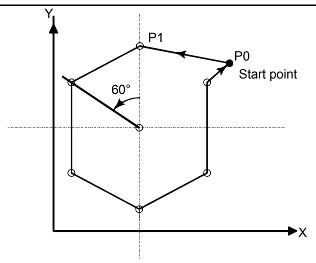
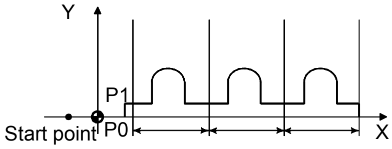
(6/10)

Functions	Illustration	Program format
Tool offset (G45 to G48)		• **For machining center** $\left\{\begin{array}{l} G45 \\ G46 \\ G47 \\ G48 \end{array}\right\}$ IP_ D_ ; D : Tool offset number
Scaling (G50, G51)		• **For machining center** G51 X_ Y_ Z_ $\left\{\begin{array}{l} P\_ \\ I\_ J\_ K\_ \end{array}\right\}$; P, I, J, K : Scaling magnification X, Y, Z : Control position of scaling G50 : Cancel • **For lathe only** Enabled when G code system B/C is used
Programmable mirror image (G50.1, G51.1)		G51.1 IP_ ; G50.1 ; . . . Cancel
Coordinate system setting Maximum spindle speed clamp (G50)		• **For lathe only** G50 IP_ ; (Coordinate system setting) G50 S_ ; (Maximum spindle speed clamp)
Local coordinate system setting (G52)		G52 IP_ ;
Command in machine coordinate system (G53)		G53 IP_ ;
Tool axis direction control (G53.1)		G53.1 ; Tool axis direction control
Selection of workpiece coordinate system (G54 to G59)		$\left\{\begin{array}{l} G54 \\ \vdots \\ G59 \end{array}\right\}$ IP_ ;

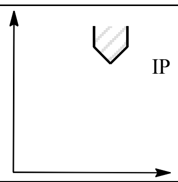
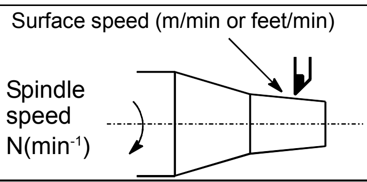
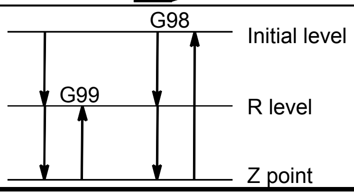
(7/10)

Functions	Illustration	Program format
Rotary table dynamic fixture offset (G54.2)		• **For machining center** G54.2 P_ ; Fixture offset P : Reference fixture offset value number G54.2 P0 ; Offset cancel
Single direction positioning (G60)		G60 IP_ ;
Cutting mode (G64) Exact stop mode (G61) Tapping mode (G63)		G64_ ; Cutting mode G61_ ; Exact stop mode G63_ ; Tapping mode
Automatic corner override (G62)		G62_ ; Automatic corner override
Custom macro (G65, G66, G66.1, G67)		One-shot call G65 P_ L_ <Argument assignment> ; P : Program number L : Number of repetition Modal call G66 P_ L_ <Argument assignment> ; Call after the move command G66.1 P_ L_ <Argument assignment> ; Each block call G67 ; Cancel
Mirror image for double turret (G68, G69)		• **For lathe only** G68 : Mirror image for double turret G69 : Mirror image cancel
Coordinate system rotation, Three-dimensional coordinate conversion (G68, G69) (G68.1, G69.1)	In case of X-Y plane	• **For machining center** $G68 \left\{\begin{matrix} G17\ X\_\ Y\_ \\ G18\ Z\_\ X\_ \\ G19\ Y\_\ Z\_ \end{matrix}\right\} R\underline{\alpha}\ ;$ G69 ; Cancel • **For lathe** $G68.1 \left\{\begin{matrix} G17\ X\_\ Y\_ \\ G18\ Z\_\ X\_ \\ G19\ Y\_\ Z\_ \end{matrix}\right\} R\underline{\alpha}\ ;$ G69.1 ; Cancel

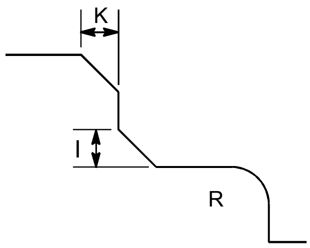
(8/10)

Functions	Illustration	Program format
Feature coordinate system selection (G68.2)		G68.2 X_ Y_ Z_ I_ J_ K_ ; Feature coordinate system setting G69 ; Feature coordinate system setting cancel X, Y, Z : Feature coordinate system origin I, J, K : Euler angles for determining the orientation of the feature coordinate system
Figure copy (G72.1, G72.2)	 	Rotational copy {(G17) (G18) (G19)} G72.1 P_ L_ {X_ Y_ / Z_ X_ / Y_ Z_} R_ ; Linear copy {(G17) (G18) (G19)} G72.2 P_ L_ {I_ J_ / K_ I_ / J_ K_} ;
Canned cycle for drilling (G73, G74, G80 to G89)		• For machining center G80 ; Cancel G73 G74 G76 G81 : G89 } X_Y_Z_P_Q_R_F_K_ ;
Canned cycle (G71 to G76) (G90, G92, G94)		**• For lathe only** N_ G70P_ Q_ ; G71U_ R_ ; G71P_ Q_ U_ W_ F_ S_ T_ ; G72W_ R_ ; G72P_ Q_ U_ W_ F_ S_ T_ ; G73U_ W_ R_ ; G73P_ Q_ U_ W_ F_ S_ T_ ; G74R_ ; G74X(u)_ Z(w)_ P_ Q_ R_ F_ ; G75R_ ; G75X(u)_ Z(w)_ P_ Q_ R_ F_ ; G76P_ Q_ R_ ; G76X(u)_ Z(w)_ P_ Q_ R_ F_ ; {G90 / G92} X_ Z_ I_ F_ ; G94X_ Z_ K_ F_ ;

(9/10)

Functions	Illustration	Program format
Absolute/incremental programming (G90/G91)		**• For machining center** G90_ ; Absolute programming G91_ ; Incremental programming : G90_ G91_ ; Programming in both modes **• For lathe** X_ Z_ C_ : Absolute programming U_ W_ H_ : Incremental programming Distinguished by an address specified in combined use with a G function such as G00 and G01.
Maximum incremental command value check (G91.1)		G91.1 IP_ ; IP_ ; Maximum incremental value Set 0 to cancel maximum incremental value check.
Change of workpiece coordinate system (G92) Maximum spindle speed clamp (G92)		**• For machining center** G92 IP_ ; Change of workpiece coordinate system G92 S_ ; Constant surface speed control : Maximum spindle speed clamp
Workpiece coordinate system preset (G92.1)		**• For machining center** G92.1 IP 0 ;
Inverse time feed (G93)		G93 ; Inverse time setting mode
Feed per minute, Feed per revolution (G94, G95) (G98, G99)	mm/min inch/min mm/rev inch/rev	**• For machining center** G94 F_ ; Feed per minute G95 F_ ; Feed per revolution **• For lathe** G98 F_ ; Feed per minute G99 F_ ; Feed per revolution
Constant surface speed control (G96, G97)		G96 S_ ; Constant surface speed control on (surface speed specification) G97 S_ ; Constant surface speed control off (spindle speed specification)
Initial point return / R point return (G98, G99)		**• For machining center** G98_ ; G99_ ;

(10/10)

Functions	Illustration	Program format
Optional chamfering/corner R Chamfering/corner R	 	• **For machining center** ,C_ : Chamfering ,R_ : Corner R • **For lathe only** C±K X_ ; P_; R_ C±K Z_ ; P_; R_

D RANGE OF COMMAND VALUE

Linear axis

- In case of millimeter input, feed screw is millimeter

	Increment system				
	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Least command increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Max. programmable dimension (mm)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999
Max. rapid traverse (mm/min)*1	999,000	999,000	100,000	10,000	1,000
Feedrate range (mm/min)*1	0.01 to 999,000	0.01 to 999,000	0.0001 to 100,000	0.00001 to 10,000	0.000001 to 1,000
Incremental feed (mm/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001
Tool compensation amount (mm)*2	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999	0 to ±9,999.99999	0 to ±999.999999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

- In case of inch input, feed screw is millimeter

	Increment system				
	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Least command increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Max. programmable dimension (inch)	±39,370.078	±39,370.0787	±3,937.00787	±393.700787	±39.3700787
Max. rapid traverse (mm/min)*1	999,000	999,000	100,000	10,000	1,000
Feedrate range (inch/min)*1	0.001 to 96,000	0.0001 to 9,600	0.00001 to 4,000	0.000001 to 400	0.0000001 to 40
Incremental feed (inch/step)	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001	0.0000001 0.000001 0.00001 0.0001
Tool compensation amount (inch)*2	0 to ±999.999	0 to ±999.9999	0 to ±999.99999	0 to ±999.999999	0 to ±99.9999999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

- In case of inch input, feed screw is inch

	Increment system				
	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Least command increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Max. programmable dimension (inch)	±99,999.999	±99,999.9999	±9,999.99999	±999.999999	±99.9999999
Max. rapid traverse (inch/min)*1	96,000	9,600	4,000	400	40
Feedrate range (inch/min)*1	0.001 to 96,000	0.0001 to 9,600	0.00001 to 4,000	0.000001 to 400	0.0000001 to 40
Incremental feed (inch/step)	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001	0.0000001 0.000001 0.00001 0.0001
Tool compensation amount (inch)*4	0 to ±999.999	0 to ±999.9999	0 to ±999.99999	0 to ±999.999999	0 to ±99.9999999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

- In case of millimeter input, feed screw is inch

	Increment system				
	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Least command increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Max. programmable dimension (mm)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999
Max. rapid traverse (inch/min)*1	96,000	9,600	4,000	400	40
Feedrate range (mm/min)*1	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000	0.00001 to 10,000	0.000001 to 1,000
Incremental feed (mm/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001
Tool compensation amount (mm)*2	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999	0 to ±9,999.99999	0 to ±999.999999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

- Rotary axis

	Increment system				
	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Least command increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Max. programmable dimension (deg)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999
Max. rapid traverse (deg/min)*1	999,000	999,000	100,000	10,000	1,000
Feedrate range (deg/min)*1	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000	0.00001 to 10,000	0.000001 to 1,000
Incremental feed (deg/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001
Tool compensation amount (deg)*2	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999	0 to ±9,999.99999	0 to ±999.999999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

NOTE

*1 The feedrate range shown above are limitations depending on CNC interpolation capacity. As a whole system, limitations depending on servo system must also be considered.

*2 Value applied when the offset extension function is used.
If the mode of input is switched between inch input and metric input, the maximum compensation value that can be set at inch input time is (maximum compensation value) × 1/25.4. If a value exceeding this value is specified at inch input time, the compensation value is not converted to a metric value correctly when the mode of input is switched to metric input.

*3 The unit is the detection unit.

*4 Depends on the increment system of the axis at in address X.

E NOMOGRAPHS

E.1 INCORRECT THREADED LENGTH

The leads of a thread are generally incorrect in δ_1 and δ_2, as shown in Fig. E.1 (a), due to automatic acceleration and deceleration.

Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.

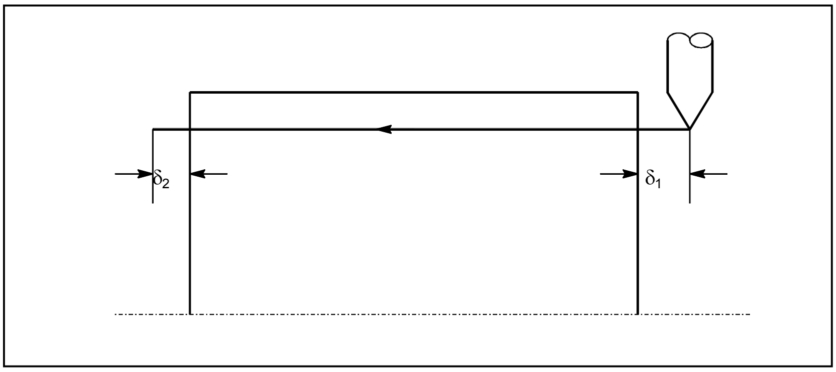


Fig. E.1 (a) Incorrect thread position

Explanation

- How to determine δ_2

$\delta_2 = T_1 V$ (mm). . . . (1)

$V = \frac{1}{60} RL$

T_1 : Time constant of servo system (sec)
V : Threading speed (mm/sec)
R : Spindle speed (min^{-1})
L : Thread feed (mm)
Time constant T_1 (sec) of the servo system: Usually 0.033 s.

- How to determine δ_1

$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\}V$ . . . . . (2)

$a = \exp(-\frac{t}{T_1})$ . . . . . (3)

T_1 : Time constant of servo system (sec)
V : Threading speed (mm/sec)
Time constant T_1 (sec) of the servo system: Usually 0.033 s.

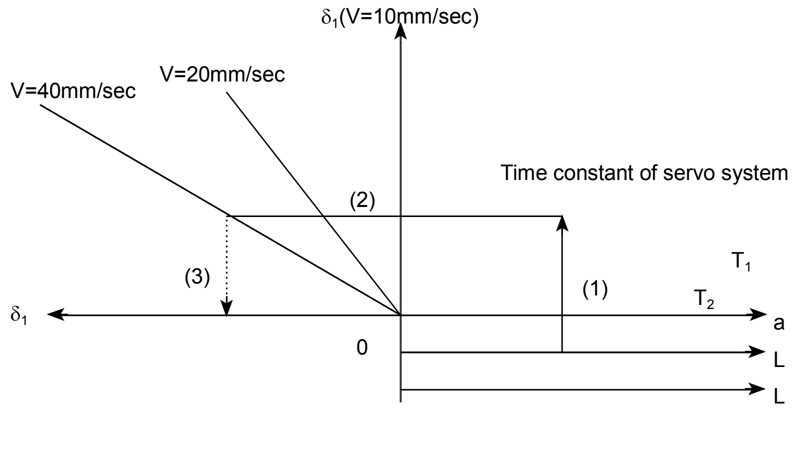
The lead at the beginning of thread cutting is shorter than the specified lead L, and the allowable lead error is ΔL. Then as follows.

$a = \frac{\Delta L}{L}$

When the value of "a" is determined, the time lapse until the thread accuracy is attained. The time "t" is substituted in (2) to determine δ_1: Constants V and T_1 are determined in the same way as for δ_2. Since the calculation of δ_1 is rather complex, a nomography is provided on the following pages.

- How to use nomograph

First specify the class and the lead of a thread. The thread accuracy, a, will be obtained at (1), and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm/s will be obtained at (2). Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/s can be obtained at (3).



(Note) See the graph in reference later in the manual for an actual example.

Fig. E.1 (b) Nomograph

> **NOTE**
> The equations for δ_1, and δ_2 are for when the acceleration/ deceleration time constant for cutting feed is 0.

E.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

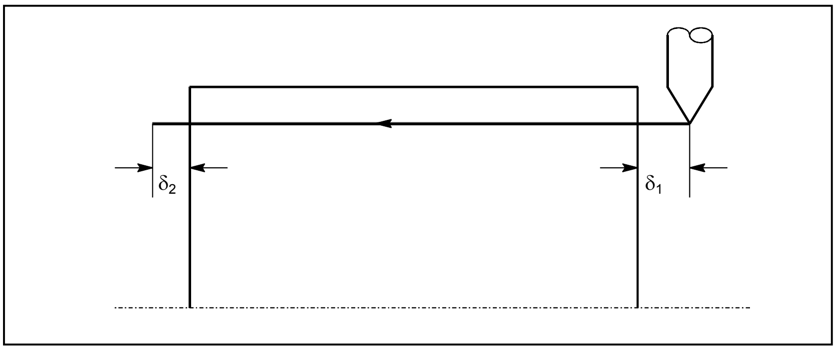


Fig. E.2 (a) Incorrect threaded portion

Explanation

- How to determine δ_2

$\delta_2 = \frac{LR}{1800*}$(mm)
R : Spindle speed (min^{-1})
L : Thread lead (mm)
* When time constant T_1 of the servo system is 0.033 s.

- How to determine δ_1

$\delta_1 = \frac{LR}{1800*}(- 1 - \ln a)$(mm)
$= \delta_2(- 1 - \ln a)$(mm)
R : Spindle speed (min^{-1})
L : Thread lead (mm)
* When time constant T_1 of the servo system is 0.033 s.
Following a is a permited value of thread.

a	-1-lna
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

Example

R=350rpm
L=1mm
a=0.01
then

$$\delta_2 = \frac{350\times1}{1800} = 0.194\text{(mm)}$$

$$\delta_1 = \delta_2\times3.605 = 0.701\text{(mm)}$$

Reference

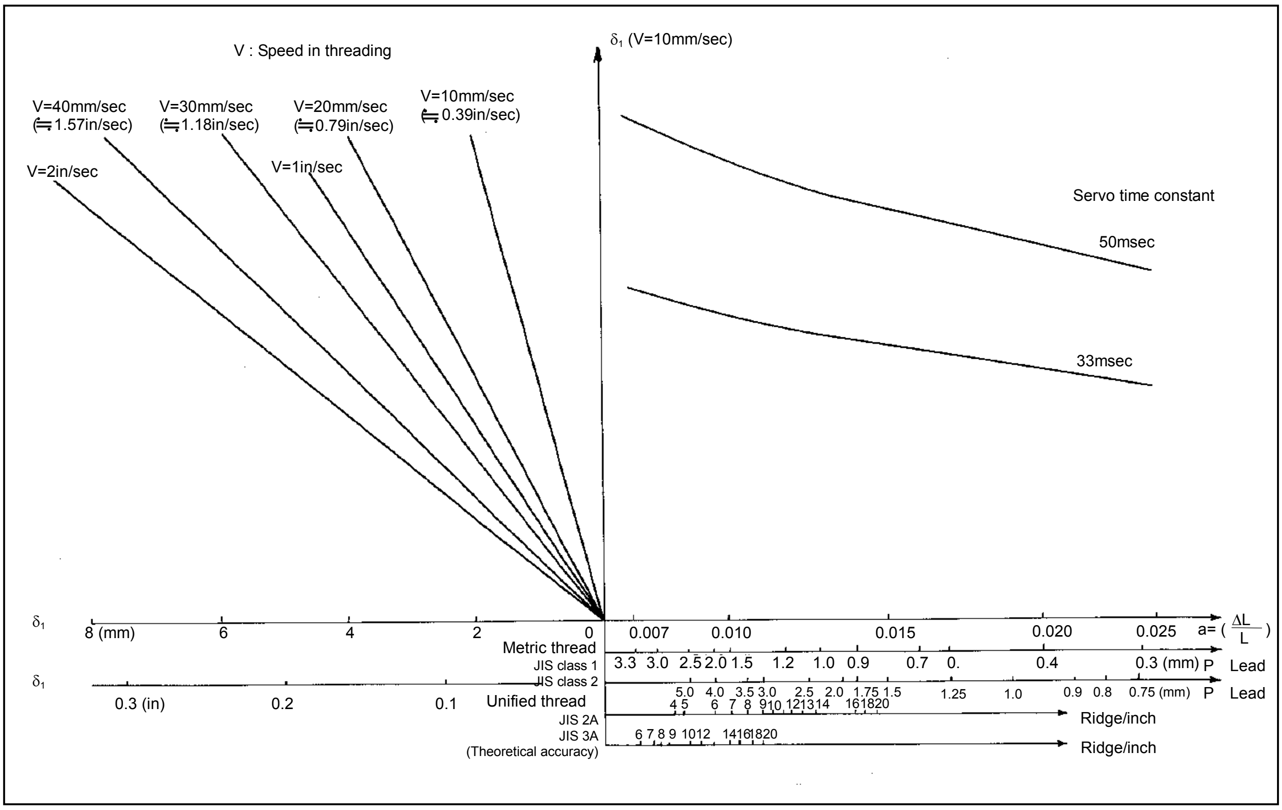


Fig. E.2 (b) Nomograph for obtaining approach distance δ_1

E.3 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. E.3 (a).

Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

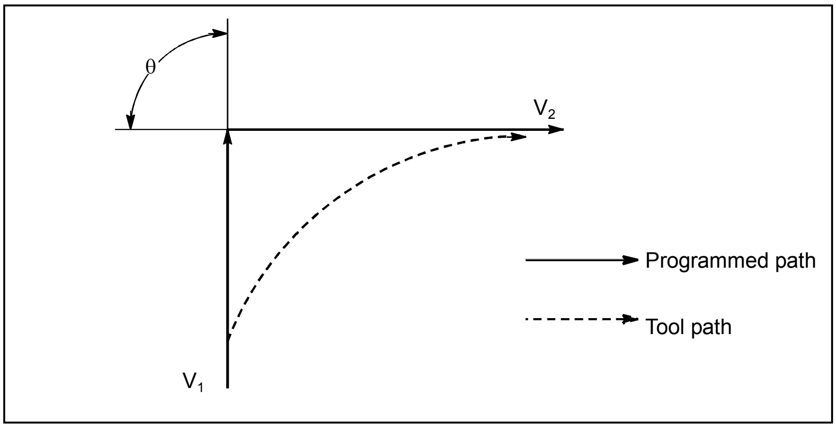


Fig. E.3 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:
- Feedrate (V_1, V_2)
- Corner angle (θ)
- Exponential acceleration / deceleration time constant (T_1) at cutting ($T_1 = 0$)
- Presence or absence of buffer register.

The above parameters are used to theoretically analyze the tool path and above tool path is drawn with the parameter which is set as an example.

When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.

In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

Explanation
- Analysis

The tool path shown in Fig. E.3 (b) is analyzed based on the following conditions:

- Feedrate is constant at both blocks before and after cornering.
- The controller has a buffer register. (The error differs with the reading speed of the tape reader, number of characters of the next block, etc.)

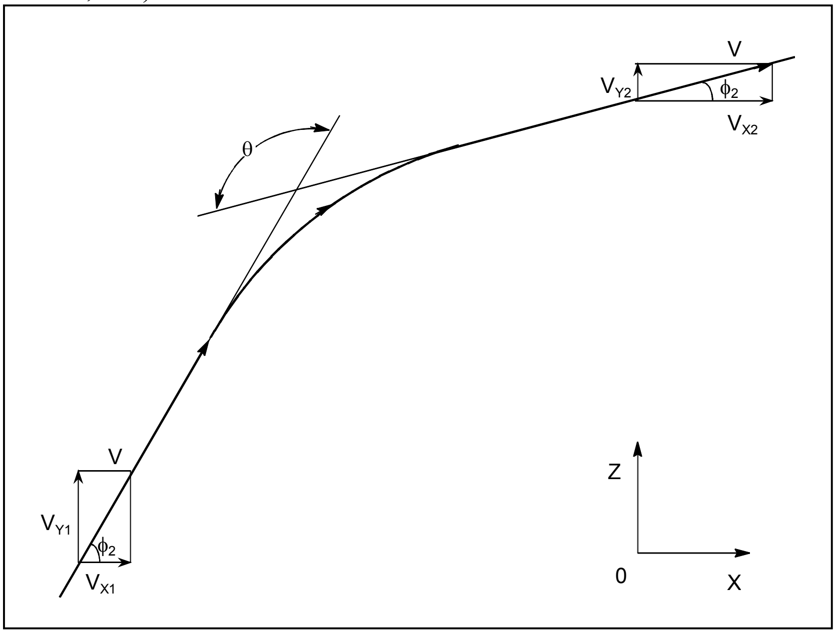


Fig. E.3 (b) Example of tool path

- Description of conditions and symbols

$V_{X1}=V\cos\phi_1$
$V_{Y1}=V\sin\phi_1$
$V_{X2}=V\cos\phi_2$
$V_{Y2}=V\sin\phi_2$

V : Feedrate at both blocks before and after cornering
V_{X1} : X-axis component of feedrate of preceding block
V_{Y1} : Y-axis component of feedrate of preceding block
V_{X2} : X-axis component of feedrate of following block
V_{Y2} : Y-axis component of feedrate of following block
θ : Corner angle
ϕ_1 : Angle formed by specified path direction of preceding block and X-axis
ϕ_2 : Angle formed by specified path direction of following block and X-axis

- Initial value calculation

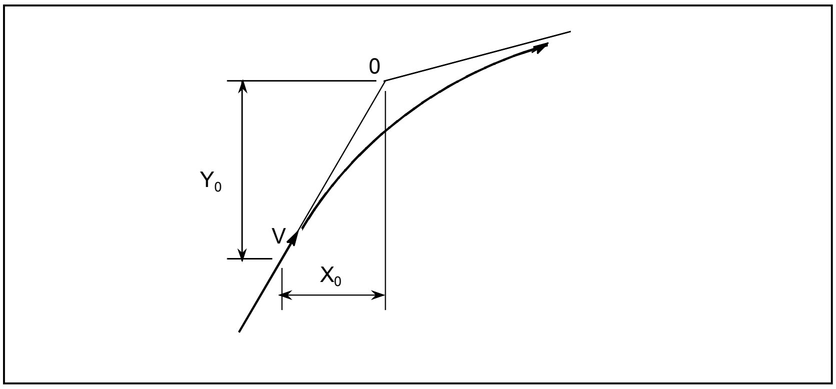


Fig. E.3 (c) Initial value

The initial value when cornering begins, that is, the X and Y coordinates at the end of command distribution by the controller, is determined by the feedrate and the positioning system time constant of the servo motor.

$$X_0=V_{X1}(T_1+T_2)$$
$$Y_0=V_{Y1}(T_1+T_2)$$

T_1: Exponential acceleration/deceleration time constant. (T=0)
T_2: Time constant of positioning system (Inverse of position loop gain)

- Analysis of corner tool path

The equations below represent the feedrate for the corner section in X-axis direction and Y-axis direction.

$$V_X(t) = (V_{X2} - V_{X1})[1 - \frac{V_{X1}}{T_1 - T_2}\{T_1\exp(- \frac{t}{T_1}) - T_2\exp(- \frac{t}{T_2})\}+V_{X1}]$$

$$= V_{x2}[1 - \frac{V_{x1}}{T_1 - T_2}\{T_1\exp(- \frac{t}{T_1}) - T_2\exp(- \frac{t}{T_2})\}]$$

$$= V_Y(t)=\frac{V_{Y1} - V_{Y2}}{T_1 - T_2}\{T_1\exp(- \frac{t}{T_1}) - T_2\exp(- \frac{t}{T_2})\}+V_{Y2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t)=\int_0^t V_X(t)dt - X_0$$

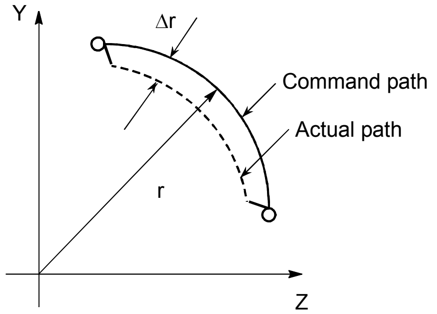
$$= \frac{V_{X2} - V_{X1}}{T_1 - T^2}\{T_1^2\exp(- \frac{t}{T_1}) - T_2^2\exp(- \frac{t}{T_2})\} - V_{X2}(T_1 + T_2 - t)$$

$$Y(t) = \int_0^t V_Y(t)dt - Y_0$$

$$= \frac{V_{Y2} - V_{Y1}}{T_1 - T_2}\{T_1^2\exp(- \frac{t}{T_1}) - T_2^2\exp(- \frac{t}{T_2})\} - V_{Y2}(T_1+T_2 - t)$$

E.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING

When a servo motor is used, the positioning system causes an error between input commands and output results. Since the tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, sepecially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2(1 - a^2)) - \frac{V^2}{r} \; . \; . \; . \; . \; . \; (1)$$

Δr: Maximum radius error (mm)
v : Feedrate (mm/sec)
r : Circle radius (mm)
T_1: Exponential acceleration/deceleration time constantat cutting (sec) (T=0)
T_2: Time constant of positoning system (sec) (Inverse of positon loop gain)
α : Feed forward coefficient (%)

In the case of bell-shaped acceleration/deceleration and linear acceleration/deceleration after cutting feed interpolation, an approximation of this radius error can be obtained with the following expression:

Linear acceleration/deceleration after cutting feed interpolation

$$\Delta r = (\frac{1}{24}T_1^2 + \frac{1}{2}T_2^2(1 - a^2)) - \frac{V^2}{r}$$

Bell-shaped acceleration/deceleration after cutting feed interpolation

$$\Delta r = (\frac{1}{48}T_1^2 + \frac{1}{2}T_2^2(1 - a^2)) - \frac{V^2}{r}$$

Thus, the radius error in the case of bell-shaped acceleration/deceleration and linear acceleration/deceleration after interpolation is smaller than in case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo loop time constant.

Since the machining radius r (mm) and allowable error Δr (mm) of the workpiece is given in actual machining, the allowable limit feedrate v (mm /sec) is determined by equation (1).
Since the acceleration/deceleration time constant at cutting which is set by this equipment varies with the machine tool, refer to the manual issued by the machine tool builder.

F CHARACTER-TO-CODES CORRESPONDENCE TABLE

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		”	034	Quotation mark
H	072		#	035	Sharp
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		’	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		-	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	At mark
1	049		[	091	Left square bracket
2	050		]	093	Right square bracket
3	051		^	094	
4	052		_	095	Underscore
5	053				

G ALARM LIST

(1) Alarms on program and operation (PS alarm)
(2) Background edit alarms (BG alarm)
(3) Communication alarms (SR alarm)

Alarm numbers are common to all these alarm types.
Depending on the state, an alarm is displayed as in the following examples:
PS"alarm number" Example: PS0003
BG"alarm number" Example: BG0085
SR"alarm number" Example: SR0001

Number	Message	Description
0001	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
0002	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting TVC parameter No. 0000#0 to “0”.
0003	TOO MANY DIGIT	Data entered with more digits than permitted in the NC instruction word. The number of permissible digits varies according to the function and the word.
0004	INVALID BREAK POINT OF WORDS	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0005	NO DATA AFTER ADDRESS	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0006	ILLEGAL USE OF MINUS SIGN	A minus sign (–) was specified at an NC instruction word or system variable where no minus signal may be specified.
0007	ILLEGAL USE OF DECIMAL POINT	A decimal point (.) was specified at an address where no decimal point may be specified, or two decimal points were specified.
0009	IMPROPER NC-ADDRESS	An illegal address was specified, or parameter 1020 is not set.
0010	IMPROPER G-CODE	An unusable G code is specified.
0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead.
0014	CAN NOT COMMAND G95	A synchronous feed is specified without the option for threading / synchronous feed. Modify the program.
0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either add on the simultaneous axis control extension option, or divide the number of programmed move axes into two blocks.

Number	Message	Description
0020	OVER TOLERANCE OF RADIUS	An arc was specified for which the difference in the radius at the start and end points exceeds the value set in parameter No. 2410. Check arc center codes I, J and K in the program. The tool path when parameter No. 2410 is set to a large value is spiral.
0021	ILLEGAL PLANE SELECT	The plane selection instructions G17 to G19 are in error. Reprogram so that same 3 basic parallel axes are not specified simultaneously. This alarm is also generated when an axis that should not be specified for plane machining is specified, for example, for circular interpolation or involute interpolation. To enable programming of 3 or more axes, the helical interpolation option must be added to each of the relevant axes.
0022	R OR I,J,K COMMAND NOT FOUND	The command for circular interpolation lacks arc radius R or coordinate I, J, or K of the distance between the start point to the center of the arc.
0025	CIRCLE CUT IN RAPID (F0)	F0 (rapid traverse in inverse feed or feed specified by an F code with 1–digit number) was specified during circular interpolation (G02, G03) or involute interpolation (G02.2, G03.2).
0027	NO AXES COMMANDED IN G43/G44	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Multiple axes were specified for the same block when the tool length compensation type is C.
0028	ILLEGAL PLANE SELECT	The plane selection instructions G17 to G19 are in error. Reprogram so that same 3 basic parallel axes are not specified simultaneously. This alarm is also generated when an axis that should not be specified for plane machining is specified, for example, for circular interpolation or involute interpolation. To enable programming of 3 or more axes, the helical interpolation option must be added to each of the relevant axes.
0029	ILLEGAL OFFSET VALUE	Illegal offset No.
0030	ILLEGAL OFFSET NUMBER	An illegal offset No. was specified. This alarm is also generated when the tool shape offset No. exceeds the maximum number of tool offset sets in the case of tool offset memory B.
0031	ILLEGAL P COMMAND IN G10	The relevant data input or option could not be found for the L No. of G10. No data setting address such as P or R was specified. An address command not concerned with data setting was specified. An address varies with the L No. The sign or decimal point of the specified address is in error, or the specified address is out of range.
0032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
0033	NO INTERSECTION AT CUTTER COMPENSATION	The intersection cannot be obtained by the intersection calculation in cutter or tool-nose radius compensation. Modify the program.
0034	NO CIRC ALLOWED IN STUP/EXT BLK	In cutter or tool-nose radius compensation, a startup or cancellation is performed in the G02 or G03 mode. Modify the program.

Number	Message	Description
0035	CAN NOT COMMANDED G31	- G31 cannot be specified. This alarm is generated when a G code (such as for cutter or tool-nose radius compensation) of group 07 is not canceled. - A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like. Or, specify the torque limit override by address Q.
0037	CAN NOT CHANGE PLANE IN G41/G42	The compensation plane G17/G18/G19 was changed during cutter or tool-nose radius compensation. Modify the program.
0038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program.
0039	CHF/CNR NOT ALLOWED IN G41,G42	Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in G41 and G42 commands (cutter compensation or tool nose radius compensation). The program may cause overcutting to occur in chamfering or corner R. Modify the program.
0041	INTERFERENCE IN CUTTER COMPENSATION	In cutter or tool-nose radius compensation, excessive cutting may occur. Modify the program.
0042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in tool compensation or three-dimensional cutter compensation. Modify the program.
0043	ILLEGAL T-CODE COMMAND	On a system with a DRILL-MATE ATC installed, M06 is not specified in a block that specifies a T code. Alternatively, a T code beyond the allowable range is specified.
0044	G27-G30 NOT ALLOWED IN FIXED CYC	One of G27 to G30 is commanded in canned cycle mode. Modify the program.
0045	ADDRESS Q NOT FOUND (G73/G83)	In a high-speed peck drilling cycle (G73) or peck drilling cycle (G83), the amount of each-time cutting is not specified by address Q, or Q0 is specified. Modify the program.
0046	ILLEGAL REFERENCE RETURN COMMAND	A command for a return to the second, third or fourth reference position is error. (The address P command is in error.) Although an option for a return to the third or fourth reference position was not set, 3 or 4 was specified in address P.
0047	ILLEGAL AXIS SELECT	Two or more parallel axes (in parallel with a basic axis) have been specified upon start-up of three-dimensional tool compensation or three-dimensional coordinate conversion.
0048	BASIC 3 AXIS NOT FOUND	Start-up of three-dimensional tool compensation or three-dimensional coordinate conversion has been attempted, but the three basic axes used when Xp, Yp, or Zp is omitted are not set in parameter No. 1022.
0049	ILLEGAL COMMAND(G68,G69)	When three-dimensional coordinate conversion (G68 or G69) was specified, the tool compensation was not canceled. Or, programs of three-dimensional coordinate conversion (G68, G69) and tool compensation (G43, G44 or G49) were not nested. Or, the three-dimensional coordinate conversion was specified during the tool length compensation and another tool length compensation was specified.
0050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
0051	MISSING MOVE AFTER CNR/CHF	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
0052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01 (or vertical line). Modify the program.

Number	Message	Description
0053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, J, K and R are specified.
0054	NO TAPER ALLOWED AFTER CHF/CNR	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
0055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount. Modify the program.
0056	NO END POINT & ANGLE IN CHF/CNR	In direct dimension drawing programming, both an end point and an angle were specified in the block next to the block in which only an angle was specified (Aa). Modify the program.
0057	NO SOLUTION OF BLOCK END	Block end point is not calculated correctly in direct dimension drawing programming. Modify the program.
0058	END POINT NOT FOUND	Block end point is not found in direct dimension drawing programming. Modify the program.
0060	SEQUENCE NUMBER NOT FOUND	[External data input/output] The specified number could not be found for program number and sequence number searches. Although input/output of a pot number of tool data or offset input was requested, no tool number was input after power on. The tool data corresponding to the entered tool number could not be found. [External workpiece number search] The program corresponding to the specified workpiece number could not be found. [Program restart] In the program restart sequence number specification, the specified sequence number could not be found.
0061	P OR Q COMMAND IS NOT IN THE MULTIPLE REPETIVE CYCLES BLOCK	Address P or Q is not specified in multiple repetitive cycle (G70, G71, G72, or G73) command.
0062	THE CUTTING AMOUNT IS ILLEGAL IN THE ROUGH CUTTING CYCLE	A zero or a negative value was specified in a multiple repetitive canned rough-cutting cycle (G71 or G72) as the depth of cut.
0063	THE BLOCK OF A SPECIFIED SEQUENCE NUMBER IS NOT FOUND	The sequence number specified by addresses P and Q in multiple repetitive cycle (G70, G71, G72, or G73) command cannot be searched.
0064	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)	In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command for the first plane axis was not a monotonous increase or decrease.
0065	G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM	In the first block of the shape program specified by P of the multiple repetitive canned cycle (G70, G71, G72, or G73), G00 or G01 was not specified.
0066	UNAVAILABLE COMMAND IS IN THE MULTIPLE REPETIVE CYCLES BLOCK	An unavailable command was found in a multiple repetitive canned cycle (G70, G71, G72, or G73) command block.
0067	THE MULTIPLE REPETIVE CYCLES IS NOT IN THE PART PROGRAM STORAGE	A multiple repetitive canned cycle (G70, G71, G72, or G73) command is not registered in a tape memory area.
0069	LAST BLOCK OF SHAPE PROGRAM IS AN ILLEGAL COMMAND	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for the chamfering or corner R in the last block is terminated in the middle.
0070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.

Number	Message	Description
0071	DATA NOT FOUND	- The address to be searched was not found. - The program with specified program number was not found in program number search. - In the program restart block number specification, the specified block number could not be found. Check the data.
0072	DATA NOT FOUND	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option) or 1000 (option). Delete unnecessary programs and execute program registration again.
0073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
0074	PROGRAM NUMBER ALREADY IN USE	The program number is other than 1 to 9999. Modify the program number.
0075	PROTECT	An attempt was made to register a program whose number was protected. In program matching, the password for the encoded program was not correct.
0076	PROGRAM NOT FOUND	The specified program is not found in the subprogram call, macro call or graphic copy. The M, G, T or S codes are called by a P instruction other than that in an M98, G65, G66, G66.1 or interrupt type custom macro, and a program is called by a No. 2 auxiliary function code. This alarm is also generated when a program is not found by these calls.
0077	TOO MANY SUB,MACRO NESTING	The total number of subprogram and macro calls exceeds the permissible range. Another subprogram call was executed during an external memory subprogram call.
0078	SEQUENCE NUMBER NOT FOUND	The specified sequence No. was not found during sequence number search. The sequence No. specified as the jump destination in GOTO— and M99P— was not found.
0079	PROGRAM NOT MATCH	The program in memory does not match the program stored on tape. Multiple programs cannot be matched continuously when parameter No. 2200#3 is set to “1”. Set parameter No. 2200#3 to “0” before executing a match.
0080	G37 MEASURING POSITION REACHED SIGNAL IS NOT PROPERLY INPUT	- For machining center series When the tool length measurement function (G37) is performed, a measuring position reached signal goes 1 in front of the area determined by the ε value specified in parameter No.6254. Alternatively, the signal does not go 1. - For lathe When the automatic tool compensation function (G36, G37) is used, a measuring position reached signals (XAE1, XAE2) does not go 1 within the range determined by the ε value specified in parameters No.6254 and No.6255.

Number	Message	Description
0081	G37 OFFSET NO. UNASSIGNED	- For machining center series The tool length measurement function (G37) is specified without specifying an H code. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified without specifying an T code. Correct the program.
0082	G37 SPECIFIED WITH H CODE	- For machining center series The tool length measurement function (G37) is specified together with an H code in the same block. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified together with an T code in the same block. Correct the program.
0083	G37 IMPROPER AXIS COMMAND	- For machining center series An error has been found in axis specification of the tool length measurement function (G37). Alternatively, a move command is specified as an incremental command. Correct the program. - For lathe An error has been found in axis specification of the automatic tool compensation function (G36, G37). Alternatively, a command is specified as an incremental command. Correct the program.
0085	OVERRUN ERROR	The next character was received from the I/O device connected to reader/punch interface 1 before it could read a previously received character.
0086	DR OFF	During I/O process by reader/punch interface 1, the data set ready input signal of the I/O device (DR) was OFF. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
0087	BUFFER OVERFLOW	During a read by reader/punch interface 1, although a read stop command was issued, more than 10 characters were input. The I/O device or printed circuit board was defective.
0090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2. An attempt was made to set the zero position for the absolute position detector by return to the reference position when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
0091	MANUAL REFERENCE POSITION RETURN IS NOT PERFORMED IN FEED HOLD	Manual return to the reference position cannot be performed when automatic operation is halted. Perform the manual return to the reference position when automatic operation is stopped or reset.
0092	ZERO RETURN CHECK (G27) ERROR	The axis specified in G27 has not returned to zero. Reprogram so that the axis returns to zero.

Number	Message	Description
0094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the User's manual.
0095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.) Perform the correct operation according to the User's manual.
0096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.) Perform the correct operation according to the User's manual.
0097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or alarms 0094 to 0097 reset, no automatic operation is performed.) Perform automatic operation.
0098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
0099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
0101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. If this alarm has occurred, press <RESET> while pressing <PROG>, and only the program being edited will be deleted. Register the deleted program.
0109	FORMAT ERROR IN G08	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
0110	OVERFLOW :INTEGER	An integer went out of range during arithmetic calculations.
0111	OVERFLOW :FLOATING	A decimal point (floating point number format data) went out of range during arithmetic calculations.
0112	ZERO DIVIDE	An attempt was made to divide by zero in a custom macro.
0113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
0114	ILLEGAL EXPRESSION FORMAT	The format used in an expression in a custom macro statement is in error. The parameter tape format is in error.
0115	VARIABLE NO. OUT OF RANGE	A number that cannot be used for a local variable, common variable, or system variable in a custom macro is specified. In the EGB axis skip function (G31.8), a non-existent custom macro variable number is specified. Or, the number of custom macro variables used to store skip positions is not sufficient.
0116	WRITE PROTECTED VARIABLE	An attempt was made in a custom macro to use on the left side of an expression a variable that can only be used on the right side of an expression.
0118	TOO MANY BRACKET NESTING	Too many brackets “[ ]” were nested in a custom macro. The nesting level including function brackets is 5.
0119	ARGUMENT VALUE OUT OF RANGE	The value of an argument in a custom macro function is out of range.
0122	TOO MANY MACRO NESTING	Too many macro calls were nested in a custom macro.
0123	ILLEGAL MODE FOR GOTO/WHILE/DO	A GOTO statement or WHILE–DO statement was found in the main program in the MDI or DNC mode.
0124	MISSING END STATEMENT	The END instruction corresponding to the DO instruction was missing in a custom macro.

Number	Message	Description
0125	MACRO STATEMENT FORMAT ERROR	The format used in a macro statement in a custom macro is in error.
0126	ILLEGAL LOOP NUMBER	DO and END Nos. in a custom macro are in error, or exceed the permissible range (valid range: 1 to 3).
0127	DUPLICATE NC,MACRO STATEMENT	An NC statement and macro statement were specified in the same block.
0128	ILLEGAL MACRO SEQUENCE NUMBER	The specified sequence No. could not be found for sequence number search. The sequence No. specified as the jump destination in GOTO-- and M99P-- could not be found.
0129	USE 'G' AS ARGUMENT	G is used as an argument in a custom macro call. G can be specified as an argument only in an every-block call (G66.1).
0130	NC AND PMC AXIS ARE CONFLICTED	The NC command and the PMC axis control command were conflicted. Modify the program or ladder.
0136	SPOS AXIS - OTHER AXIS SAME TIME	The spindle positioning axis and another axis are specified in the same block.
0137	M-CODE & MOVE CMD IN SAME BLK.	The spindle positioning axis and another axis are specified in the same block.
0138	SUPERIMPOSED DATA OVERFLOW	The total distribution amount of the CNC and PMC is too large during superimposed control for PMC axis control.
0139	CANNOT CHANGE PMC CONTROL AXIS	The PMC axis was selected for the axis for which the PMC axis is being controlled.
0140	PROGRAM NUMBER ALREADY IN USE	In the background, an attempt was made to select or delete the program being selected in the foreground. Perform the correct operation for the background edition.
0141	CAN NOT COMMAND G51 IN 3-D OFFSET	G51 (Scaling ON) is commanded in the three-dimensional tool offset mode. Modify the program.
0142	ILLEGAL SCALE RATE	The scaling rate is 0 times or 10000 times or more. Modify the setting of the scaling rate. (G51P_ … or G51I_J_K_ … or parameter (No. 5411 or 5421))
0143	COMMAND DATA OVERFLOW	An overflow occurred in the storage length of the CNC internal data. This alarm is also generated when the result of internal calculation of scaling, coordinate rotation and cylindrical interpolation overflows the data storage. It also is generated during input of the manual intervention amount.
0144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.
0145	ILLEGAL USE OF G12.1/G13.1	The axis No. of plane selection parameter No. 5460 (linear axis) and No. 5461(axis of rotation) in the polar coordinate interpolation mode is out of range (1 to number of controlled axes).
0146	ILLEGAL USE OF G-CODE	The modal G code group contains an illegal G code in the polar coordinate interpolation mode or when a mode was canceled. Only the following G codes are allowed: G40, G50, G69.1 An illegal G code was specified while in the polar coordinate interpolation mode. The following C codes are not allowed: G27, G28, G30, G30.1, G31 to G31.4, G37 to G387.3, G52, G92, G53, G17 to G19, G81 to G89, G68 In the 01 group, G codes other than G01, G02, G03, G02.2 and G03.2 cannot be specified.

Number	Message	Description
0148	SETTING ERROR	Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714).
0154	NOT USING TOOL IN LIFE GROUP	H99 or D99 is specified when no tool management data number is assigned to the spindle position. Correct the program.
0160	MISMATCH WAITING M-CODE	A waiting M-code is in error. <1> When different M codes are specified for path 1 and path 2 as waiting M codes without a P command. <2> When the waiting M codes are not identical even though the P commands are identical <3> When the waiting M codes are identical and the P commands are not identical (This occurs when a P command is specified with binary value.) <4> When the number lists in the P commands contain a different number even though the waiting M codes are identical (This occurs when a P command is specified by combining path numbers.) <5> When a waiting M code without a P command (2-path waiting) and a waiting M code with a P command (3-or-more-path waiting) were specified at the same time <6> When a waiting M code without a P command was specified for 3 or more paths.
0161	ILLEGAL P OF WAITING M-CODE	P in a waiting M-code is incorrect. <1> When address P is negative <2> When a P value inappropriate for the system configuration was specified <3> When a waiting M code without a P command (2-path waiting) was specified in the system having 3 or more paths.
0163	ILLEGAL COMMAND IN G68/G69	G68 and G69 are not independently commanded in balance cut. An illegal value is commanded in a balance cut combination (address P).
0169	ILLEGAL TOOL GEOMETRY DATA	Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.
0175	ILLEGAL G07.1 AXIS	An axis which cannot perform cylindrical interpolation was specified. More than one axis was specified in a G07.1 block. An attempt was made to cancel cylindrical interpolation for an axis that was not in the cylindrical interpolation mode. For the cylindrical interpolation axis, set not “0” but one of 5, 6 or 7 (parallel axis specification) to parameter No. 1022 to instruct the arc with axis of rotation (ROT parameter No. 1006#1 is set to “1” and parameter No. 1260 is set) ON.
0176	ILLEGAL G-CODE USE(G07.1 MODE)	A G code was specified that cannot be specified in the cylindrical interpolation mode. This alarm also is generated when an 01 group G code was in the G00 mode or code G00 was instructed. Cancel the cylindrical interpolation mode before instructing code G00.
0179	PARAM. (NO.7510) SETTING ERROR	The number of controlled axes set by the parameter No. 7510 exceeds the maximum number. Modify the parameter setting value.
0190	ILLEGAL AXIS SELECTED (G96)	An illegal value was specified in P in a G96 block or parameter No. 5844.

Number	Message	Description
0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode or simple spindle synchronous control mode.
0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.
0199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
0200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value was out of range or was not specified. The parameter (Nos. 5241 to 5243) setting is an S value which can be specified for the rigid tap. Correct the parameters or modify the program.
0201	FEEDRATE NOT FOUND IN RIGID TAP	The command F code for a cutting feedrate is a zero. If the value of F command is much smaller than that of the S command, when a rigid tap command is specified, this alarm is generated. This is because cutting is not possible by the lead specified by the program.
0202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. (System error)
0203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
0204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (or G74) block. Modify the program.
0205	RIGID MODE DI SIGNAL OFF	Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (DGN G061.0) is not ON during execution of the G84 (or G74) block. Check the PMC ladder diagram to find the reason why the DI signal is not turned on.
0206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Modify the program.
0207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
0210	CAN NOT COMMAND M198/M99	1 The execution of an M198 or M99 command was attempted during scheduled operation. Alternatively, the execution of an M198 command was attempted during DNC operation. Modify the program. 2 The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle.
0212	ILLEGAL PLANE SELECT	The direct drawing dimensions programming is commanded for the plane other than the Z-X plane. Correct the program.
0213	ILLEGAL COMMAND IN SYNCHRO-MODE	In feed axis synchronization control, the following errors occurred during the synchronous operation. 1) The program issued the move command to the slave axis. 2) The program issued the manual operation to the slave axis. 3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.
0214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
0217	DUPLICATE G51.2(COMMANDS)	G51.2 is further commanded in the G51.2 mode. Modify the program.

Number	Message	Description
0218	NOT FOUND P/Q COMMAND	P or Q is not commanded in the G51.2 block, or the command value is out of the range. Modify the program. For a polygon turning between spindles, more information as to why this alarm occurred is indicated in DGN No. 471.
0219	COMMAND G51.2/G50.2 INDEPENDENTLY	G51.2 and 50.2 were specified in the same block for other commands. Modify the program in another block.
0220	ILLEGAL COMMAND IN SYNCHR-MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis. Modify the program or check the PMC ladder.
0221	ILLEGAL COMMAND IN SYNCHR-MODE	Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program.
0222	DNC OP. NOT ALLOWED IN BG-EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.
0224	ZERO RETURN NOT FINISHED	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when the parameter ZRNx (No. 1005#0) is set to 0.
0231	ILLEGAL FORMAT IN G10 L52	Errors occurred in the specified format at the programmable-parameter input.
0232	TOO MANY HELICAL AXIS COMMAND	Three or more axes were specified as helical axes in the helical interpolation mode. Five or more axes were specified as helical axes in the helical interpolation B mode.
0233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS-232-C interface, other users were using it.
0239	BP/S ALARM	While punching was being performed with the function for controlling external I/O units ,background editing was performed.
0240	BP/S ALARM	Background editing was performed during MDI operation.
0241	ILLEGAL FORMAT IN G02.2/G03.2	The end point of an involute curve on the currently selected plane, or the center coordinate instruction I, J or K of the corresponding basic circle, or basic circle radius R was not specified.
0242	ILLEGAL COMMAND IN G02.2/G03.2	An illegal value was specified in the involute curve. The coordinate instruction I, J or K of the basic circle on the currently selected plane or the basic circle radius R is “0”, or the start and end points are not inside the basic circle.
0243	OVER TOLERANCE OF END POINT	The end point is not positioned on the involute curve that passes through the start point, and this error exceeds the permissible error limit (parameter No. 2510).
0244	P/S ALARM	In torque control, the total permissible move value specified as a parameter is exceeded.
0245	T-CODE NOT ALLOWED IN THIS BLOCK	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.
0247	THE MISTAKE IS FOUND IN THE OUTPUT CODE OF DATA.	When an encrypted program is output, EIA is set for the output code. Specify ISO.
0250	TOOL CHANGE ILLEGAL Z AXIS COMMAND	A Z-axis move command was performed in the same block for M06 command.
0251	TOOL CHANGE ILLEGAL T COMMAND	An unusable T code was specified in M06Txx.
0253	G05 CAN NOT BE COMMANDED	A binary operation was specified during advanced preview control mode.

Number	Message	Description
0300	ILLEGAL COMMAND IN SCALING	An illegal G code was specified during scaling. Modify the program. For the T system, one of the following functions is specified during scaling, this alarm is generated. - finishing cycle (G70 or G72) - outer surface rough-cutting cycle (G71 or G73) - end side rough-cutting cycle (G72 or G74) - closed loop cutting cycle (G73 or G75) - end side cutting-off cycle (G74 or G76) - outer surface or inner surface cutting-off cycle (G75 or G77) - multiple repetitive threading cycle (G76 or G78) - face drill cycle (G83 or G83) - face tap cycle (G84 or G84) - face boring cycle (G85 or G85) - side drill cycle (G87 or G87) - side tap cycle (G88 or G88) - side boring cycle (G89 or G89) - outer surface turning cycle or inner surface boring cycle (G77 or G20) - threading cycle (G78 or G21) - end side turning cycle (G79 or G24) (Specify G codes for systems B and C in that order.)
0301	RESETTING OF REFERENCE RETURN IS INHIBITED	Although parameter No. 1012#0 (IDGx) was set to 1 to inhibit the reference position from being set again for a return to the reference position without a dog, an attempt was made to perform a manual return to the reference position.
0302	SETTING THE REFERENCE POSITION WITHOUT DOG IS NOT PERFORMED	The reference position could not be set for a return to the reference position without a dog. Possible causes are: - The axis was not moved in the direction of a return to the reference position for jog feeding. - The axis was moved in the direction opposite to the direction of a manual return to the reference position.
0303	REFERENCE POSITION RETURN IS NOT PERFORMED	When the setting of a reference position at any position was possible in Cs contour control (parameter CRF (No. 3700#0) = 1), a G00 command was issued for the Cs contour axis without a return to the reference position after the serial spindle was switched to Cs contour control mode. Perform a reference position return with a G28 command before issuing a G00 command.
0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
0305	INTERMEDIATE POSITION IS NOT ASSIGNED	Although a G28 (automatic return to the reference position), G30 (return to the second, third, or fourth reference position), or G30/1 (return to the floating reference position) command was not issued after power-up, G29 (return from the reference position) was commanded.
0306	MISMATCH AXIS WITH CNR/CHF	The correspondence between the moving axis and the I, J, or K command is incorrect in a block in which chamfering is specified.
0307	CAN NOT START REFERENCE RETURN WITH MECHANICAL STOPPER SETTING	An attempt was made to set a butt-type reference position for an axis for which to use the function to set a reference position without a dog.
0308	G72.1 NESTING ERROR	G72.1 was specified again during G72.1 rotation copying.
0309	G72.2 NESTING ERROR	G72.2 was specified again during G72.2 parallel copying.

Number	Message	Description
0310	FILE NOT FOUND	The specified file could not be found during a subprogram or macro call.
0311	CALLED BY FILE NAME FORMAT ERROR	An invalid format was specified to call a subprogram or macro using a file name.
0312	ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING	Direct input of drawing dimensions was commanded in an invalid format. An attempt was made to specify an invalid G code during direct input of drawing dimensions. Two or more blocks not to be moved exist in consecutive commands that specify direct input of drawing dimensions. Although non-use of commas (,) (parameter No. 3405#4 = 1) was specified for direct input of drawing dimensions, a comma was specified.
0313	ILLEGAL LEAD COMMAND	The variable-lead threading increment specified in address K exceeds the specified maximum value in variable-lead threading. Or, a negative lead value was specified.
0314	ILLEGAL SETTING OF POLYGONAL AXIS	An axis was specified invalidly in polygon turning. For polygon turning: A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning between spindles: Valid spindles are not specified. (Parameter Nos. 7640 to 7643) - A spindle other than the serial spindle. - A spindle is not connected.
0315	ILLEGAL NOSE ANGLE COMMAND IS IN THE THREAD CUTTING CYCLE	An invalid tool tip angle is specified in a multiple repetitive canned threading cycle (G76).
0316	ILLEGAL CUTTING AMOUNT IS IN THE THREAD CUTTING CYCLE	An minimum depth of cut higher than the thread height is specified in a multiple repetitive canned threading cycle (G76).
0317	ILLEGAL THREAD COMMAND IS IN THE THREAD CUTTING CYCLE	A zero or a negative value is specified in a multiple repetitive canned threading cycle (G76) as the thread height or the depth of cut.
0318	ILLEGAL RELIEF AMOUNT IS IN THE DRILLING CYCLE	Although an escape directions is set in a multiple repetitive canned cutting-off cycle (G74 or G75), a negative value is specified for Δd.
0319	THE END POINT COMMAND IS ILLEGAL IN THE DRILLING CYCLE	Although the Δi or Δk travel distance is set to 0 in a multiple repetitive canned cutting-off cycle (G74 or G75), a value other than 0 us specified for a U or W.
0320	ILLEGAL MOVEMENT AMOUNT/CUTTING AMOUNT IS IN THE DRILLING CYCLE	A negative value is specified in a multiple repetitive canned cutting-off cycle (G74 or G75) as Δi or Δk (travel distance/the depth of cut).
0321	ILLEGAL REPEATED TIME IS IN THE PATTERN REPEATING CYCLE	A zero or a negative value is specified in a multiple repetitive canned closed loop cycle (G73) as a repeated time.
0322	FINISHING SHAPE WHICH OVER OF STARTING POINT	An invalid shape which is over the cycle starting point is specified in a shape program for a multiple repetitive canned rough-cutting cycle (G71 or G72).
0323	THE FIRST BLOCK OF SHAPE PROGRAM IS A COMMAND OF TYPE II	Type II is specified in the first block of the shape program specified by P in a multiple repetitive canned rough-cutting cycle (G71 or G72). Z (W) command is for G71. X (U) command is for G72.
0324	THE INTERRUPTION TYPE MACRO WAS DONE IN THE MULTIPLE REPETIVE CYCLES	An interruption type macro was issued during the multiple repetitive canned cycle (G70, G71, G72, or G73).

Number	Message	Description
0325	UNAVAILABLE COMMAND IS IN SHAPE PROGRAM	An usable command was issued in a shape program for a multiple repetitive canned cycle (G70, G71, G72, or G73).
0326	LAST BLOCK OF SHAPE PROGRAM IS A DIRECT DRAWING DIMENSIONS	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for direct input of drawing dimensions in the last block is terminated in the middle.
0327	MODAL THAT MULTIPLE REPETIVE CYCLES CANNOT BE DONE	A multiple repetitive canned cycle (G70, G71, G72, or G73) was commanded in a modal state in which a multiple repetitive canned cycle could not be commanded.
0328	ILLEGAL WORK POSITION IS IN THE TOOL NOSE RADIUS COMPENSATION	The specification for the blank side for a tool-nose radius compensation (G41 or G42) is incorrect in a multiple repetitive canned cycle (G71 or G72).
0329	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)	In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command of the second plane axis was not a monotonous increase or decrease.
0330	ILLEGAL AXIS COMMAND IS IN THE TURNING CANNED CYCLE	An axis other than the plane is specified n a canned cycle(G90, G92, or G94).
0331	ILLEGAL AXIS NUMBER IN AX[]	An illegal value is specified for an AX[] axis number.
0332	ILLEGAL AXIS ADDRESS IN AXNUM[]	An illegal value is specified for an AXNUM[] axis address.
0333	TOO MANY SPINDLE COMMANDS	Multiple spindle commands could be found in the same block in using an expansion spindle name. Only one spindle could be commanded in the same block.
0334	OFFSET IS OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (malfunction prevention function)
0335	PLURAL M CODE	Multiple M codes are commanded simultaneously in a block for a wait function with peripheral devices by an M code.
0336	TOOL COMPENSATION COMMANDED MORE TWO AXES	For a tool length compensation C, an attempt was made to command the offset to other axes without canceling the offset. Or, for a tool length compensation C, multiple axes are specified in G43 or G44 block.
0337	EXCESS MAXIMUM INCREMENTAL VALUE	The command value exceeded the maximum amount of incremental. (malfunction prevention function)
0338	CHECK SUM ERROR	An incorrect value was detected in a check sum. (malfunction prevention function)
0340	ILLEGAL RESTART(NANO SMOOTHING)	With manual absolute turned on, an attempt was made to restart the operation in nano smoothing mode after performing the manual interaction.
0341	TOO MANY COMMAND BLOCK (NANO SMOOTHING)	There are more blocks than can be commanded consecutively in nano smoothing mode.
0342	CUSTOM MACRO INTERRUPT ENABLE IN NANO SMOOTHING	A custom macro interrupt was enabled in nano smoothing mode. Or, nano smoothing mode was commanded with a custom macro interrupt enabled.
0343	ILLEGAL COMMAND IN NANO SMOOTHING	G43, G44, or G49 was commanded during a nano smoothing.
0344	CANNOT CONTINUE NANO SMOOTHING	An illegal command or operation by which a nano smoothing could not be continued was performed.
0345	TOOL CHANGE ILLEGAL Z AXIS POS	A tool change position on the Z-axis is incorrect.
0346	TOOL CHANGE ILLEGAL TOOL NUM	A tool change position is not set.
0347	TOOL CHANGE ILLEGAL COMMAND IN SAME BLK.	Tool changing is commanded twice or more in the same block.

Number	Message	Description
0348	TOOL CHANGE Z AXIS POS NOT ESTABLISHED	A tool change spindle on the Z-axis is not set.
0349	TOOL CHANGE SPINDLE NOT STOP	A tool change spindle stop is not stopped.
0350	PARAMETER OF THE INDEX OF THE SYNCHRONOUS CONTROL AXIS SET ERROR.	An illegal synchronization control axis number (parameter No. 8180) is set.
0351	BECAUSE THE AXIS IS MOVING, THE SYNC CONTROL IS CAN'T BE USED.	While the axis being subject to synchronization control was moving, an attempt was made to start or cancel the synchronization control by a synchronization control axis selection signal.
0352	SYNCHRONOUS CONTROL AXIS COMPOSITION ERROR.	This error occurred when: 1) An attempt was made to perform synchronization control for the axis during a synchronization, composition, or superposition. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation. 3) An attempt was made to operate synchronization control although a parent-child-grandchild relation was not set.
0353	THE INSTRUCTION WAS DONE FOR THE AXIS WHICH WAS NOT ABLE TO MOVE.	This error occurred when: - For synchronization 1) A move command was issued to the axis for which parameter No. 8163#7NUMx is set to 1. 2) A move command was issued to the slave axis. - For composition 1) A move command was issued to the axis for which parameter No. 8163#7NUMx is set to 1. 2) A move command was issued to the axis for which parameter No. 8162#7MUMx is set to 1.
0354	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN SYNC MODE	This error occurred when G28 was specified to the master axis being parking during synchronization control, but an axis reference position is not set for the slave axis.
0355	PARAMETER OF THE INDEX OF THE COMPOSITE CONTROL AXIS SET ERROR.	An illegal composite control axis number (parameter No. 8183) is specified.
0356	BECAUSE THE AXIS IS MOVING, THE COMP CONTROL IS CAN'T BE USED.	While the axis being subject to composite control was moving, an attempt was made to start or cancel the composite control by a composite control axis selection signal.
0357	COMPOSITE CONTROL AXIS COMPOSITION ERROR.	This error occurred when an attempt was made to perform composite control for the axis during a synchronization, composition, or superposition.
0359	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN COMP MODE	This error occurred when G28 was specified to the composite axis during composite control, but a reference position is not set to the other part of the composition.
0360	PARAMETER OF THE INDEX OF THE SUPERPOS CONTROL AXIS SET ERROR.	An illegal superposition control axis number (parameter No. 8186) is specified.
0361	BECAUSE THE AXIS IS MOVING, THE SUPERPOS CONTROL IS CAN'T BE USED.	While the axis being subject to superposition control was moving, an attempt was made to start or cancel the superposition control by a superposition control axis selection signal.

Number	Message	Description
0362	SUPERPOSITION CONTROL AXIS COMPOSITION ERROR.	This error occurred when: 1) An attempt was made to perform superposition control for the axis during a synchronization, composition, or superposition. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation.
0363	THE G28 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	This error occurred when G28 was specified to the superposition control slave axis during superposition control.
0364	THE G53 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	This error occurred when G53 was specified to the slave axis being moved during superposition control.
0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The maximum control axis number or maximum control spindle number which could be used within a path was exceeded. (For a loader path, this alarm is generated if the number of axis per path is set to 5 or greater.)
0366	IMPROPER G-CODE IN TURRET METHOD	When the turret change tools method was selected (parameter No. 5040#3 (TCT) = 0), G43, G43.1, G43.4, G43.5, or G43.7 was commanded.
0367	3-D CONV. WAS COMMANDED IN SYNC MODE AS THE PARAMETER PKUx(NO.8162#2) IS 0.	A three-dimensional coordinate conversion was commanded during synchronization control when the parameter PKUx (No.8162#2) was 0.
0368	OFFSET REMAIN AT OFFSET COMMAND	When the ATC change tools method was selected (parameter No. 5040#3 (TCT) = 1) during G43, G43.1, G43.4, or G43.5 mode, G43.7 was commanded. Or, G43, G43.1, G43.4, or G43.5 was commanded during G43.7 mode.
0369	G31 FORMAT ERROR	- No axis is specified or tow or more axes are specified in the torque limit switch instruction (G31P98/P99). - The specified torque Q value in the torque limit switch instruction is out of range. The torque Q range is 1 to 99.
0370	G31P/G04Q ERROR	The specified address P value for G31 is out of range. The address P range is 1 to 4 in a multistage skip function. The specified address Q value for G04 is out of range. The address Q range is 1 to 4 in a multistage skip function. Or, P1-4 for G31, or Q1-4 for G04 was commanded without a multistage skip function option.
0371	ILLEGAL FORMAT IN G10 OR L50	In a command format for a programmable parameter input, an attempt was made to change the parameter for an encryption (No. 3220), key (No. 3221), or protection range (No.3222 or No.3223) as a "the encryption function for the key and program." Modify the program.
0372	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the orthogonal axis.
0373	ILLEGAL HIGH-SPEED SKIP SIGNAL	In the skip commands (G31, G31P1 to G31P4) and dwell commands (G04, G04Q1 to G04Q4), the same high-speed signal is selected in different paths.

Number	Message	Description
0374	ILLEGAL REGISTRATION OF TOOL MANAGER(G10)	G10L75 or G10L76 data was registered during the following data registration: - From the PMC window. - From the FOCAS2. - By G10L75 or G10L76 in another system. Command G10L75 or G10L76 again after the above operation is completed.
0375	CAN NOT ANGULAR CONTROL(SYNC:MIX:OVL)	Angular axis control is disabled for this axis configuration. 1) When some related axes under angular axis control are not in synchronous control mode or when one angular axis is not paired with the other angular axis or one Cartesian axis is not paired with the other Cartesian axis in synchronous control 2) When some related axes under composite control are not in composite control mode or when one angular axis is not paired with the other angular axis or one Cartesian axis is not paired with the other Cartesian axis in composite control 3) When related axes under angular axis control is switched to superposition control mode1)
0376	SERIAL DCL: ILLEGAL PARAMETER	1. When Parameter No.1815#1 is set to “1”, parameter No.2002#3 is set to “0” 2. The absolute-position detection function is enabled. (Parameter No.1815#5 is set to “1”.)
0387	ILLEGAL RTM DI/DO VAR	There is no DI/DO variable that has a specified signal address (alphabet, number).
0389	ILLEGAL RTM SIGNAL BIT	Bits other than bits 0 to 7 cannot be specified with a DI/DO signal.
0391	RTM BRANCH OVER	The number of branches supported with real time custom macros was exceeded.
0392	TOO MANY SENTENCE CONTROL	Many reserved words (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, G65, M99) for RTM control were used in a real time macro command.
0393	NO SENTENCE CONTROL	In a real time macro command, there is no data to be assigned.
0394	ILLEGAL SENTENCE CONTROL	The matching of reserved words (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, G65, M99) for RTM control is incorrect.
0395	ILLEGAL NC WORD CONTROL	Control code G65 or M99 for calling a subprogram or returning from a subprogram is not coded correctly.
0396	ILLEGAL RTM SENTENCE CONTROL	In other than a real time macro command, a reserved word (ZONCE, ZEDGE, ZWHILE, ZDO, or ZEND) for RTM control is used.
0397	RTM BUFFER OVER	There is no buffer available for real time macro commands. Too many blocks read in advance are buffered as triggers used by real time macro commands.
0398	'ID OVER IN BUFFER	In blocks read in advance, there are too many real time macro commands with the same ID.
0399	'ID EXECUTION IN SAME TIME	An attempt was made to execute real time macro commands with the same ID by using the same NC statement as a trigger.
0400	ONESHOT CMDOVER	Too many one-shot real time macro commands are specified.
0401	EXEC CMD NUM OVER IN SAME TIME	The number of real time macro commands that can be executed simultaneously was exceeded
0402	ILLEGAL TOKEN FOR RTM	A token, variable, or function that is not supported by the real time custom macro function was detected.
0403	ACCESS TO RTM PROTECT VAR	An attempt was made to access a protected variable.

Number	Message	Description
0404	RTM ERROR	An error related to a real time macro command occurred.
0406	CODE AREA SHORTAGE	The storage size of the real time macro area is insufficient.
0407	DOULE SLASH IN RTM MODE	In the compile mode, an attempt was made to set the compile mode again.
0408	G90 IS NOT PERMITTED	The absolute command cannot be specified.
0409	ILLEGAL AXIS NO	An invalid axis number is specified.
0410	MIDDLE POINT IS NOT ZERO	An intermediate point other than 0 is specified with G28.
0411	SIMULTANEOUSLY AXES OVER	The maximum number of axes that can be controlled simultaneously was exceeded.
0412	ILLEGAL G CODE	An unusable G code was used.
0413	ILLEGAL ADDRESS	An unusable address was used.
0414	ILLEGAL PMC AXIS NO.	An invalid PMC axis number is specified.
0415	GROUP IS IN USE	The group to which the specified axis belongs is already in used.
0416	UNABLE TO USE THE AXIS	The specified axis cannot be used.
0417	AXIS IS UNABLE TO MOVE	The specified axis is placed in the inoperative state.
0418	ILLEGAL FEED SETTING	An incorrect feedrate is set.
0419	ILLEGAL DISTANCE SETTING	A travel distance beyond the specifiable range is specified.
0420	CONSTANT NUMBER P	A subprogram is specified not by using a constant.
0421	ILLEGAL ARGUMENT G54	With G65, an invalid argument, L, is used.
0422	ILLEGAL ARGUMENT G54	With G65, an invalid argument is used.
0423	NO PMC AXIS CONTROL OPTION	The option for PMC axis control is missing.
0424	MULTIPLE AXES IN ONE GROUP	Multiple axes are using one group.
0425	ONE AXIS USE MULTIPLE GROU	One axis is using multiple groups.
0429	ILLEGAL COMMAND IN G10.6	When retract was started in a threading block, a retract command had been issued for the long axis direction of threading.
1014	ILLEGAL FORMAT OF PROGRAM NO.	Address O or N is not followed by a number.
1016	EOB NOT FOUND	EOB (End of Block) code is missing at the end of a program input in the MDI mode.
1018	M99 IN MAIN PROGRAM	A M99 was commanded during main program when the parameter AMM (No. 7712#4) = 1 was set.
1059	COMMAND IN BUFFERING MODE	The manual intervention compensation request signal MIGET became “1” when a advanced block was found during automatic operation. To input the manual intervention compensation during automatic operation, a sequence for manipulating the manual intervention compensation request signal MIGET is required in an M code instruction without buffering.
1077	PROGRAM IN USE	An attempt was made in the foreground to execute a program being edited in the background. The currently edited program cannot be executed, so end editing and restart program execution.
1079	PROGRAM FILE NOT FOUND	The program of the specified file No. is not registered in an external device. (external device subprogram call)
1080	DUPLICATE DEVICE SUB PROGRAM CALL	Another external device subprogram call was made from a subprogram after the subprogram called by the external device subprogram call.
1081	EXT DEVICE SUB PROGRAM CALL MODE ERROR	The external device subprogram call is not possible in this mode.

Number	Message	Description
1091	DUPLICATE SUB-CALL WORD	More than one subprogram call instruction was specified in the same block.
1092	DUPLICATE MACRO-CALL WORD	More than one macro call instruction was specified in the same block.
1093	DUPLICATE NC-WORD & M99	An address other than O, N, P or L was specified in the same block as M99 during the macro modal call state.
1095	TOO MANY TYPE-2 ARGUMENT	More than ten sets of I, J and K arguments were specified in the type–II arguments (A, B, C, I, J, K, I, J, K, ...) for custom macros.
1096	ILLEGAL VARIABLE NAME	An illegal variable name was specified. A code that cannot be specified as a variable name was specified. [#_OFSxx] does not match the tool offset memory option configuration.
1097	TOO LONG VARIABLE NAME	The specified variable name is too long.
1098	NO VARIABLE NAME	The specified variable name cannot be used as it is not registered.
1099	ILLLEGAL SUFFIX [ ]	A suffix was not specified to a variable name that required a suffix enclosed by [ ]. A suffix was specified to a variable name that did not require a suffix enclosed by [ ]. The value enclosed by the specified [ ] was out of range.
1100	CANCEL WITHOUT MODAL CALL	Call mode cancel (G67) was specified even though macro continuous–state call mode (G66) was not in effect.
1101	ILLEGAL CNC STATEMENT IRT.	An interrupt was made in a state where a custom macro interrupt containing a move instruction could not be executed.
1115	READ PROTECTED VARIABLE	An attempt was made in a custom macro to use on the right side of an expression a variable that can only be used on the left side of an expression.
1120	ILLEGAL ARGUMENT FORMAT	The specified argument in the argument function (ATAN, POW) is in error.
1124	MISSING DO STATEMENT	The DO instruction corresponding to the END instruction was missing in a custom macro.
1125	ILLEGAL EXPRESSION FORMAT	The description of the expression in a custom macro statement contains an error. A parameter program format error. The screen displayed to enter periodic maintenance data or item selection menu (machine) data does not match the data type.
1128	SEQUENCE NUMBER OUT OF RANGE	The jump destination sequence No. in a custom macro statement GOTO instruction was out of range (valid range: 1 to 99999999).
1131	MISSING OPEN BRACKET	The number of left brackets ([) is less than the number of right brackets (]) in a custom macro statement.
1132	MISSING CLOSE BRACKET	The number of right brackets (]) is less than the number of left brackets ([) in a custom macro statement.
1133	MISSING '='	An equal sign (=) is missing in the arithmetic calculation instruction in a custom macro statement.
1134	MISSING ','	A delimiter (,) is missing in a custom macro statement.
1137	IF STATEMENT FORMAT ERROR	The format used in the IF statement in a custom macro is in error.
1138	WHILE STATEMENT FORMAT ERROR	The format used in the WHILE statement in a custom macro is in error.
1139	SETVN STATEMENT FORMAT ERROR	The format used in the SETVN statement in a custom macro is in error.

Number	Message	Description
1141	ILLEGAL CHARACTER IN VAR. NAME	The SETVN statement in a custom macro contacts a character that cannot be used in a variable name.
1142	TOO LONG V-NAME (SETVN)	The variable name used in a SETVN statement in a custom macro exceeds 8 characters.
1143	BPRNT/DPRNT STATEMENT FORMAT ERROR	The format used in the BPRINT statement or DPRINT statement is in error.
1144	G10 FORMAT ERROR	The G10 L No. contains no relevant data input or corresponding option. Data setting address P or R is not specified. An address not relating to the data setting is specified. Which address to specify varies according to the L No. The sign, decimal point or range of the specified address are in error.
1145	G10.1 TIME OUT	The response to a G10.1 instruction was not received from the PMC within the specified time limit.
1146	G10.1 FORMAT ERROR	The G10.1 instruction format is in error.
1152	G31.9/G31.8 FORMAT ERROR	The format of the G31.9 or G31.8 block is erroneous in the following cases: - The axis was not specified in the G31.9 or G31.8 block. - Multiple axes were specified in the G31.9 or G31.8 block. - The P code was specified in the G31.9 or G31.8 block.
1153	CANNOT USE G31.9	G31.9 cannot be specified in this modal state. This alarm is also generated when G31.9 is specified when a group 07 G code (e.g. cutter compensation) is not canceled.
1160	COMMAND DATA OVERFLOW	An overflow occurred in the position data within the CNC. This alarm is also generated if the target position of a command exceeds the maximum stroke as a result of calculation such as coordinate conversion, offset, or introduction of a manual intervention amount.
1180	ALL PARALLEL AXES IN PARKING	All of the axis specified for automatic operation are parked.
1196	ILLEGAL DRILLING AXIS SELECTED	An illegal axis was specified for drilling in a canned cycle for drilling. If the zero point of the drilling axis is not specified or parallel axes are specified in a block containing a G code in a canned cycle, simultaneously specify the parallel axes for the drilling axis.
1200	PULSCODER INVALID ZERO RETURN	The grid position could not be calculated during grid reference position return using the grid system as the one–revolution signal was not received before leaving the deceleration dog. This alarm is also generated when the tool does not reach a feedrate that exceeds the servo error amount preset to parameter No. 1841 before the deceleration limit switch is left (deceleration signal *DEC returns to “1”).
1202	NO F COMMAND AT G93	F codes in the inverse time specification mode (G93) are not handled as modal, and must be specified in individual blocks.
1223	ILLEGAL SPINDLE SELECT	An attempt was made to execute an instruction that uses the spindle although the spindle to be controlled has not been set correctly.
1282	ILLEGAL COMMAND IN 3-D OFFSET	An illegal G code was specified in the three–dimensional tool offset mode.
1283	ILLEGAL IJK IN 3-D OFFSET	When bit 0 (ONI) of parameter No. 6029 is set to 1, I, J, and K commands are specified without the decimal point in three–dimensional tool compensation mode.

Number	Message	Description
1298	ILLEGAL INCH/METRIC CONVERSION	An error occurred during inch/metric switching.
1300	ILLEGAL ADDRESS	The axis No. address was specified even though the parameter is not an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Axis No. cannot be specified in pitch error compensation data.
1301	MISSING ADDRESS	The axis No. was not specified even though the parameter is an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Or, data No. address N, or setting data address P or R are not specified.
1302	ILLEGAL DATA NUMBER	A non–existent data No. was found while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. An invalid address R value is specified in a pattern program for each machining purpose on the high–speed high–precision setting screen. This alarm is also generated when illegal word values are found.
1303	ILLEGAL AXIS NUMBER	An axis No. address exceeding the maximum number of controlled axes was found while loading parameters from a tape or by entry of the G10 parameter.
1304	TOO MANY DIGIT	Data with too many digits was found while loading parameters or pitch error compensation data from a tape.
1305	DATA OUT OF RANGE	Out–of–range data was found while loading parameters or pitch error compensation data from a tape. The values of the data setting addresses corresponding to L Nos. during data input by G10 was out of range. This alarm is also generated when NC programming words contain out–of–range values.
1306	MISSING AXIS NUMBER	A parameter which requires an axis to be specified was found without an axis No. (address A) while loading parameters from a tape.
1307	ILLEGAL USE OF MINUS SIGN	Data with an illegal sign was found while loading parameters or pitch error compensation data from a tape, or by entry of the G10 parameter. A sign was specified to an address that does not support the use of signs.
1308	MISSING DATA	An address not followed by a numeric value was found while loading parameters or pitch error compensation data from a tape.
1329	ILLEGAL MACHINE GROUP NUMBER	An machine group No. address exceeding the maximum number of controlled machine groups was found while loading parameters from a tape or by entry of the G10 parameter.
1330	ILLEGAL SPINDLE NUMBER	An spindle No. address exceeding the maximum number of controlled spindles was found while loading parameters from a tape or by entry of the G10 parameter.
1331	ILLEGAL PATH NUMBER	An path No. address exceeding the maximum number of controlled path was found while loading parameters from a tape or by entry of the G10 parameter.
1332	DATA WRITE LOCK ERROR	Could not load data while loading parameters, pitch error compensation data and work coordinate data from tape.
1333	DATA WRITE ERROR	Could not write data while loading data from tape.

Number	Message	Description
1360	PARAMETER OUT OF RANGE (TLAC)	Illegal parameter setting. (Set value is out of range.)
1361	PARAMTER SETTING ERROR 1 (TLAC)	Illegal parameter setting. (axis of rotation setting)
1362	PARAMETER SETTING ERROR 2 (TLAC)	Illegal parameter setting (tool axis setting)
1370	PARAMETER SETTING ERROR (DM3H-1)	Out–of–range data was set during setting of the three–dimensional handle feed parameter.
1371	PARAMETER SETTING ERROR (DM3H-2)	An illegal axis of rotation was set during setting of the three–dimensional handle feed parameter.
1372	PARAMETAR SETTING ERROR (DM3H-3)	An illegal master axis was set during setting of the three–dimensional handle feed parameter.
1373	PARAMETER SETTING ERROR (DM3H-4)	An illegal parallel axis or twin table was set during setting of the three–dimensional handle feed parameter.
1470	G40.1 –G42.1 PARAMETER MISS	A parameter setting related to normal direction control is illegal. The axis number of a normal direction controlled axis is set in parameter No. 5480, but that axis number is in the range of the number of controlled axes. The axis set as a normal direction controlled axis is not set as a rotation axis (ROTx, bit 0 of parameter No. 1006) = 1 and No.1022=0). Set the feedrate at which to insert rotation about a normal direction controlled axis in parameter No. 5481, in the range of 1 to 15000 mm/min.
1508	DUPLICATE M-CODE (INDEX TABLE REVERSING)	A function to which the same code as this M code is set exists. (index table indexing)
1509	DUPLICATE M-CODE (SPOS AXIS ORIENTATION)	A function to which the same code as this M code is set exists. (spindle positioning, orientation)
1510	DUPLICATE M-CODE (SPOS AXIS POSITIONING)	A function to which the same code as this M code is set exists. (spindle positioning, positioning)
1511	DUPLICATE M-CODE (SPOS AXIS RELEASE)	A function to which the same code as this M code is set exists. (spindle positioning, mode cancel)
1531	ILLEGAL USE OF DECIMAL POINT (F-CODE)	When the feedrate instruction contains valid data below the decimal point, the alarm is set and the F code contains valid data below the decimal point.
1532	ILLEGAL USE OF DECIMAL POINT (E-CODE)	When the feedrate instruction contains valid data below the decimal point, the alarm is set and the E code contains valid data below the decimal point.
1533	ADDRESS F UNDERFLOW (G95)	The feedrate for the hole drilling axis calculated from the F and S codes is too slow in the feed per single rotation mode (G95).
1534	ADDRESS F OVERFLOW (G95)	The feedrate for the hole drilling axis calculated from the F and S codes is too fast in the feed per single rotation mode (G95).
1535	ADDRESS E UNDERFLOW (G95)	The feedrate for the hole drilling axis calculated from the E and S codes is too slow in the feed per single rotation mode (G95).
1536	ADDRESS E OVERFLOW (G95)	The feedrate for the hole drilling axis calculated from the E and S codes is too fast in the feed per single rotation mode (G95).
1537	ADDRESS F UNDERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too slow.
1538	ADDRESS F OVERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too fast.
1539	ADDRESS E UNDERFLOW (OVERRIDE)	The speed obtained by applying override to the E instruction is too slow.
1540	ADDRESS E OVERFLOW (OVERRIDE)	The speed obtained by applying override to the E instruction is too fast.

Number	Message	Description
1541	S-CODE ZERO	“0” has been instructed as the S code.
1542	FEED ZERO (E-CODE)	“0” has been instructed as the feedrate (E code).
1543	ILLEGAL GEAR SETTING	The gear ratio between the spindle and position coder, or the set position coder number of pulses is illegal in the spindle positioning function.
1544	S-CODE OVER MAX	The S command exceeds the maximum spindle rotation number.
1548	ILLGAL AXIS MODE	The spindle positioning axis/Cs contour control axis was specified during switching of the controlled axis mode.
1561	ILLEGAL INDEXING ANGLE	The specified angle of rotation is not an integer multiple of the minimum indexing angle.
1564	INDEX TABLE AXIS – OTHER AXIS SAME TIME	The index table indexing axis and another axis have been specified in the same block.
1567	INDEX TABLE AXIS DUPLICATE AXIS COMMAND	Index table indexing was specified during axis movement or on an axis for which the index table indexing sequence was not completed.
1580	ENCODE ALARM (PSWD&KEY)	When an attempt was made to read a program, the specified password did not match the password on the tape and the password on tape was not equal to 0. When an attempt was made to punch an encrypted tape, the password was not in the range 0 to 99999999. The password parameter is No. 2210.
1581	ENCODE ALARM (PARAMETER)	When an attempt was made to punch an encrypted tape, the punch code parameter was set to EIA. Set parameter ISO (No. 0000#1) to “0”. An incorrect instruction was specified for program encryption or protection. This alarm is generated if an attempt is made to perform program editing, deletion, or range-specified punch-out in the protected range in the lock state. Or, a program outside the protected range is specified in rage specification punch-out in the unlock state. The protected range is defined from the program No. preset by parameter No. 3222 up to the program No. preset to parameter No. 3223. When both parameters are set to “0”, the protected range becomes O9000 to O9999.
1590	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
1591	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting TVC parameter No. 0000#0 to “0”.
1592	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is read. For the program restart function, this alarm is generated if a specified block is not found.

Number	Message	Description
1593	EGB PARAMETER SETTING ERROR	Error in setting a parameter related to the EGB (1) The setting of SYN, bit 0 of parameter No. 2011, is not correct. (2) The slave axis specified with G81 is not set as a rotation axis. (ROT, bit 0 of parameter No. 1006) (3) Number of pulses per rotation (Parameter (No. 7772 or No. 7773) or (No. 7782 or 7783) is not set.) (4) For a hobbing-machine-compatible command, parameter No. 7710 is not specified.
1594	EGB FORMAT ERROR	Error in the format of the block of an EGB command (1) T (number of teeth) is not specified in the G81 block. (2) In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. (3) n the G81 block, only one of P and Q is specified. (4) In the G81.5 block, there is no command for the master or slave axis. (5) In the G81.5 block, data out of the specified range is specified for the master or slave axis.
1595	ILL-COMMAND IN EGB MODE	During synchronization with the EGB, a command that must not be issued is issued. (1) Slave axis command using G27, G28, G29, G30,G30.1, G33, G53, etc. (2) Inch/metric conversion command using G20, G21, etc.
1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.
1597	EGB AUTO PHASE FORMAT ERROR	Format error in the G80 or G81 block in EGB automatic phase synchronization (1) R is outside the permissible range.
1598	EGB AUTO PHASE PARAMETER SETTING ERROR	Error in the setting of a parameter related to EGB automatic phase synchronization (1) The acceleration/deceleration parameter is not correct. (2) The automatic phase synchronization parameter is not correct.
1805	ILLEGAL COMMAND	[I/O Device] An attempt was made to specify an illegal command during I/O processing on an I/O device. [G30 Zero Return] The P address Nos. for instructing No. 2 to No. 4 zero return are each out of the range 2 to 4. [Single Rotation Dwell] The specified spindle rotation is “0” when single rotation dwell is specified. [Three–dimensional Tool Offset] A G code that cannot be specified was specified in the three–dimensional tool offset mode. Scaling instruction G51, skip cutting G31 and automatic tool length measurement G37 were specified.
1806	DEVICE TYPE MISS MATCH	An operation not possible on the I/O device that is currently selected in the setting was specified. This alarm is also generated when file rewind is instructed even though the I/O device is not a FANUC Cassette.

Number	Message	Description
1807	PARAMETER SETTING ERROR	An I/O interface option that has not yet been added on was specified. The external I/O device and baud rate, stop bit and protocol selection settings are erroneous.
1808	DEVICE DOUBLE OPENED	An attempt was made to open a device that is being accessed.
1809	ILLEGAL COMMAND IN G41/G42	Specified direction tool length compensation parameters are incorrect. A move instruction for a axis of rotation was specified in the specified direction tool length compensation mode.
1823	FRAMING ERROR(1)	The stop bit of the character received from the I/O device connected to reader/punch interface 1 was not detected.
1830	DR OFF(2)	The data set ready input signal DR of the I/O device connected to reader/punch interface 2 turned OFF.
1832	OVERRUN ERROR(2)	The next character was received from the I/O device connected to reader/punch interface 2 before it could read a previously received character.
1833	FRAMING ERROR(2)	The stop bit of the character received from the I/O device connected to reader/punch interface 2 was not detected.
1834	BUFFER OVERFLOW(2)	The NC received more than 10 characters of data from the I/O device connected to reader/punch interface 2 even though the NC sent a stop code (DC3) during data reception.
1889	ILLEGAL COMMAND IN G54.3	An illegal command was issued in G54.3 block. (1) An attempt was made to command G54.3 in a mode in which it cannot be accepted. (2) The command was not issued in a single block.
1898	ILLEGAL PARAMETER IN G54.2	An illegal parameter (Nos. 6068 to 6076) was specified for fixture offset.
1912	V-DEVICE DRIVER ERROR (OPEN)	An error occurred during device driver control.
1960	ACCESS ERROR (MEMORY CARD)	Illegal memory card accessing This alarm is also generated during reading when reading is executed up to the end of the file without detection of the EOR code.
1961	NOT READY (MEMORY CARD)	The memory card is not ready.
1962	CARD FULL (MEMORY CARD)	The memory card has run out of space.
1963	CARD PROTECTED (MEMORY CARD)	The memory card is write–protected.
1964	NOT MOUNTED (MEMORY CARD)	The memory card could not be mounted.
1965	DIRECTORY FULL (MEMORY CARD)	The file could not be generated in the root directory for the memory card.
1966	FILE NOT FOUND (MEMORY CARD)	The specified file could not be found on the memory card.
1967	FILE PROTECTED (MEMORY CARD)	The memory card is write–protected.
1968	ILLEGAL FILE NAME (MEMORY CARD)	Illegal memory card file name
1969	ILLEGAL FORMAT (MEMORY CARD)	Check the file name.
1970	ILLEGAL CARD (MEMORY CARD)	This memory card cannot be handled.
1971	ERASE ERROR (MEMORY CARD)	An error occurred during memory card erase.
1972	BATTERY LOW (MEMORY CARD)	The memory card battery is low.
1973	FILE ALREADY EXIST	A file having the same name already exists on the memory card.
1990	SPL:ILLEGAL AXIS COMMAND	The axis specified by the smooth interpolation (G5.1Q2) is illegal.
1993	SPL:CAN'T MAKE VECTOR	The end point and the 2 previous point are the same in generation of the 3–dimensional tool offset vector by the end point for smooth interpolation.

Number	Message	Description
1995	ILLEGAL PARAMETER IN G41.2/G42.2	The parameter settings (parameter Nos. 6080 to 6089) for determining the relationship between the axis of rotation and the rotation plane are incorrect.
1999	ILLEGAL PARAMETER IN G41.3	The parameter settings (parameter Nos. 6080 to 6089) for determining the relationship between the axis of rotation and the rotation plane are incorrect.
2002	NO KNOT COMMAND (NURBS)	Knot has not been specified, or a block not related to NURBS interpolation was specified in the NURBS interpolation mode.
2003	ILLEGAL AXIS COMMAND (NURBS)	An axis not specified as a control point was specified in the No. 1 block.
2004	ILLEGAL KNOT	There is an insufficient number of knot individual blocks.
2005	ILLEGAL CANCEL (NURBS)	The NURBS interpolation mode was turned OFF even though NURBS interpolation was not completed.
2006	ILLEGAL MODE (NURBS)	A mode that cannot be paired with the NURBS interpolation mode was specified.
2007	ILLEGAL MULTI-KNOT	Nested knots for each level can be specified for the start and end points.
2051	#200-#499ILLEGAL P-CODE MACRO COMMON INPUT(NO OPTION)	An attempt was made to enter a custom macro common variable not existing in the system.
2052	#500-#549P-CODE MACRO COMMON SELECT(CANNOT USE SETVN)	The variable name cannot be entered. The SETVN command cannot be used with the P-CODE macro common variables #500 to #549.
2053	THE NUMBER OF #30000 IS UNMATCH	An attempt was made to enter a P-CODE-only variable not existing in the system.
2054	THE NUMBER OF #40000 IS UNMATCH	An attempt was made to enter an extended P-CODE-only variable not existing in the system.
2060	ILLEGAL PARAMETER IN G43.4/G43.5	The parameter for the pivot tool length compensation is incorrect.
2061	ILLEGAL COMMAND IN G43.4/G43.5	An illegal command was specified in tool center point control. - A rotation axis command was specified in tool center point control (type 2) mode. - With a table rotary type or mixed-type machine, a I, J, or K command was specified in the tool center point control (type 2) command (G43.5) block. - A command that does not move the tool center point (only a rotation axis is moved) was specified for the workpiece in the G02 mode. - G43.4 or G43.5 was specified in the tool center point control mode. - When the workpiece coordinate system is set as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 is 1), G02 or G03 was specified while the rotation axis was not perpendicular to the plane.
4010	ILLEGAL REAL VALUE OF OBUF :	The real value for a output buffer is in error.
5006	TOO MANY WORD IN ONE BLOCK	The number of words in a block exceeds the maximum. The maximum is 26 words. However, this figure varies according to NC options. Divide the instruction word into two blocks.
5007	TOO LARGE DISTANCE	Due to compensation, point of intersection calculation, interpolation or similar reasons, a movement distance that exceeds the maximum permissible distance was specified. Check the programmed coordinates or compensation amounts.
5009	PARAMETER ZERO (DRY RUN)	The dry run feedrate parameter No. 1410 or maximum cutting feedrate parameter No. 1422 for each axis has been set to 0.

Number	Message	Description
5010	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is read.
5011	PARAMETER ZERO (CUT MAX)	The maximum cutting feedrate parameter No. 1430 has been set to 0.
5014	TRACE DATA NOT FOUND	A transfer could not be made because of no trace data.
5015	NO ROTATION AXIS	No rotation axis was found in a handle feed in the tool axis direction or in the tool axis right angle direction.
5016	ILLEGAL COMBINATION OF M CODES	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.
5018	POLYGON SPINDLE SPEED ERROR	In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained. For polygon turning between spindles: More information as to why this alarm occurred is indicated in DGN No. 471.
5020	PARAMETER OF RESTART ERROR	The setting of parameter No. 7310 for specifying the order of the axes on which to move to the machining restart position in a dry run is invalid. The valid range is from 1 to the number of controlled axes.
5043	TOO MANY G68 NESTING	Three-dimensional coordinate conversion has been specified three or more times. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.
5044	G68 FORMAT ERROR	Errors for three-dimensional coordinate conversion command are: (1) No I, J, or K command was issued in three-dimensional coordinate conversion command block. (without coordinate rotation option) (2) All of I, J, or K command were 0 in three-dimensional coordinate conversion command block. (3) No rotation angle R was not commanded in three-dimensional coordinate conversion command block.
5046	ILLEGAL PARAMETER (S-COMP)	The setting of a parameter related to straightness compensation contains an error. Possible causes include: - A non-existent axis number is set in a moving or compensation axis parameter. - More than 128 pitch error compensation points are set between the furthest points in the negative and position regions. - The straightness compensation point numbers do not have correct magnitude relationships. - No straightness compensation point is found between the furthest pitch error compensation point in the negative region and that in the positive region. - The compensation per compensation point is either too large or too small.
5050	ILL-COMMAND IN G81.1 MODE	During chopping, a move command has been issued for the chopping axis.

Number	Message	Description
5058	G35/G36 FORMAT ERROR	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.
5060	ILLEGAL PARAMETER IN G02.3/G03.3	The axis parameter setting to perform an exponential interpolation is in error. Parameter No. 5641: A liner axis number for performing an exponential interpolation Parameter No. 5642: A rotation axis number for performing an exponential interpolation The settable value is 1 to the number of control axes, but it must not be duplicated.
5061	ILLEGAL FORMAT IN G02.3/G03.3	The exponential interpolation command (G02.3/G03.3) has a format error. The command range for address I or J is -89.0 to -1.0 or +1.0 to +89.0. No I or J is specified or out-of -range value is specified. No address R, or 0 is specified.
5062	ILLEGAL COMMAND IN G02.3/G03.3	The value specified in an exponential interpolation command (G02.3/03.3) is illegal. A value that does not allow exponential interpolation is specified. (For example, the value for In is 0 or negative.)
5064	DIFFERRENT AXIS UNIT	Circular interpolation has been specified on a plane consisting of axes having different increment systems.
5065	DIFFERRENT AXIS UNIT(PMC AXIS)	Axes having different increment systems have been specified in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
5066	RESTART ILLEGAL SEQUENCE NUMBER	A sequence number from 7000 to 7999 was read during the search for the next number in a restart program for the back or restart function.
5068	FORMAT ERROR IN G31P90	No travel axis was specified. Two or more travel axes were specified.
5073	NO DECIMAL POINT	No decimal point has been specified for an address requiring a decimal point.
5074	ADDRESS DUPLICATION ERROR	The same address has been specified two or more times in a single block. Alternatively, two or more G codes in the same group have been specified in a single block.
5085	SMOOTH IPL ERROR 1	A block for specifying smooth interpolation contains a syntax error.
5115	ILLEGAL ORDER (NURBS)	There is an error in the specification of the rank.
5116	ILLEGAL KNOT VALUE (NURBS)	Monotone increasing of knots is not observed.
5117	ILLEGAL 1ST CONTROL POINT (NURBS)	The first control point is incorrect. Or, it does not provide a continuity from the previous block.
5118	ILLEGAL RESTART (NURBS)	After manual intervention with manual absolute mode set to on, NURBS interpolation was restarted.

Number	Message	Description
5122	ILLEGAL COMMAND IN SPIRAL	A spiral interpolation or conical interpolation command has an error. Specifically, this error is caused by one of the following: 1) L = 0 is specified. 2) Q = 0 is specified. 3) R/, R/, C is specified. 4) Zero is specified as height increment. 5) Zero is specified as height difference. 6) Three or more axes are specified as the height axes. 7) A height increment is specified when there are two height axes. 8) Q is specified when radius difference = 0. 9) Q < 0 is specified when radius difference > 0. 10) Q > 0 is specified when radius difference < 0. 11) A height increment is specified when no height axis is specified.
5123	OVER TOLERANCE OF END POINT IN SPIRAL	The difference between a specified end point and the calculated end point exceeds the allowable range (parameter 3471).
5124	CAN NOT COMMAND SPIRAL	A spiral interpolation or conical interpolation was specified in any of the following modes: 1) Scaling 2) Polar coordinate interpolation 3) In cutter compensation C mode, the center is set as the end point.
5130	NC AND SUPERIMPOSE AXIS CONFLICT	In the PMC superposition axis control, the NC command and The PMC axis control command were conflicted. Modify the program and the ladder.
5131	NC COMMAND IS NOT COMPATIBLE	The PMC axis control and three-dimensional coordinate conversion or a polar coordinate interpolation were specified simultaneously.
5132	CANNOT CHANGE SUPERIMPOSED AXIS	The superposition axis was selected for the axis for which the PMC superposition axis is being controlled.
5195	DIRECTION CAN NOT BE JUDGED	For a one–contact input touch sensor used with the tool compensation amount measurement value direct input B function, stored pulse directions are not unified. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. Alternatively, the tool is moving along two axes (X–axis and Z–axis) simultaneously.
5196	ILLEGAL AXIS OPERATION	During HPCC or during the execution of a 5-axis-related function, an unavailable function was used.
5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting parameter (No.1819#2) is set to "1". Move the machine to reference position by manual operation and execute manual reference return.
5257	G41/G42 NOT ALLOWED IN MDI MODE	Cutter compensation or tool nose radius compensation was specified in MDI mode. (Depending on the setting of the parameter MCR (No. 5008#4))
5303	TOUCH PANEL ERROR	The touch panel is not connected correctly, or the touch panel cannot be initialized when the power is turned on. Correct the cause then turn on the power again.

Number	Message	Description
5305	ILLEGAL SPINDLE NUMBER	In a spindle select function by address P for a multiple spindle control, 1) Address P is not specified. 2) Parameter No.3781 is not specified to the spindle to be selected. 3) An illegal G code which cannot be commanded with an S_P_; command is specified. 4) A multi spindle cannot be used because the parameter EMS (No. 3702#1) is 1.
5312	ILLEGAL COMMAND IN G10 L75/76/77	One of formats in G10L75, G10L76, or G10L77 to G11 commands is in error, or the command value is out of data range. Modify the program.
5316	TOOL TYPE NUMBER NOT FOUND	A tool with the specified tool-type number could not be found. Modify the program or register the tool.
5317	ALL TOOL LIFE IS OVER	The lives of all tools with the specified tool-type number have expired. Replace the tool.
5320	DIA./RAD. MODE CAN'T BE SWITCHED .	In any of the following states, diameter/radius specification was switched: 1) When a buffered program is being executed 2) When a movement is being made on the axis
5329	M98 AND NC COMMAND IN SAME BLOCK	A subprogram call which is not a single block was commanded during canned cycle mode.
5360	TOOL INTERFERENCE CHECK ERROR	This alarm is issued when interference with another tool is caused by a data modification based on G10 data input or file reading or when an attempt is made to modify the tool figure data of a tool registered in the cartridge.
5361	ILLEGAL MAGAZINE DATA	Tools stored in the cartridge are interfering with each other. Reregister the tools in the cartridge, or modify the tool management data or tool figure data. If this alarm is issued, no tool interference check is made when tools are registered in the cartridge management table. Moreover, empty pot search operation does not operate normally. If this alarm is issued, the power must be turned off before operation is continued.
5406	G41.3/G40 FORMAT ERROR	(1) The G41.3 or G40 block contains a move command. (2) The G41.3 block contains a G or M code that suppresses buffering.
5407	ILLEGAL COMMAND IN G41.3	(1) In the G41.3 mode, a G code of group 01 other than G00 and G01 is specified. (2) In the G41.3 mode, an offset command (a G code of group 07) is specified. (3) The block next to G41.3 (startup) specifies no movement.
5408	G41.3 ILLEGAL START_UP	(1) In a mode of group 01 other than G00 and G01, G41.3 (startup) is specified. (2) The included angle between the tool vector and move vector is 0 or 180 degrees at the time of startup.
5420	ILLEGAL PARAMETER IN G43.4/G43.5	A parameter related to tool center point control is illegal.

Number	Message	Description
5421	ILLEGAL COMMAND IN G43.4/G43.5	An illegal command was specified in tool center point control. - A rotation axis command was specified in tool center point control (type 2) mode. - With a table rotary type or mixed-type machine, a I,J,K command was specified in the tool center point control (type 2) command (G43.5) block. - A command that does not move the tool center point (only a rotation axis is moved) was specified for the workpiece in the G02 mode. - G43.4 or G43.5 was specified in the tool center point control mode. - When the workpiece coordinate system is set as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 is 1), G02 or G03 was specified while the rotation axis was not perpendicular to the plane.
5422	EXCESS VELOCITY IN G43.4/G43.5	An attempt was made to make a movement at an axis feedrate exceeding the maximum cutting feedrate by tool center point control.
5425	ILLEGAL OFFSET VALUE	The offset number is incorrect.
5430	ILLEGAL COMMAND IN 3-D CIR	In a modal state in which three-dimensional circular interpolation cannot be specified, a three-dimensional circular interpolation (G02.4/G03.4) is specified. Alternatively, in three-dimensional circular interpolation mode, a code that cannot be specified is specified.
5432	G02.4/G03.4 FORMAT ERROR	A three-dimensional circular interpolation command (G02.4/G03.4) is incorrect.
5433	MANUAL INTERVENTION IN G02.4/G03.4 (ABS ON)	In three-dimensional circular interpolation mode (G02.4/G03.4), manual intervention was made when the manual absolute switch was on.
5435	PARAMETER OUT OF RANGE (TLAC)	Illegal parameter setting. (Set value is out of range.)
5436	ILLEGAL PARAMETER SETTING OF ROTARY AXIS(TLAC)	Illegal parameter setting. (axis of rotation setting)
5437	ILLEGAL PARAMETER SETTING OF MASTER ROTARY AXIS(TLAC)	Illegal parameter setting. (master axis of rotation setting)
5445	CAN NOT COMMAND MOTION IN G39	Corner circular interpolation (G39) of cutter compensation or tool nose radius compensation is not specified alone but is specified with a move command.
5446	NO AVOIDANCE AT G41/G42	Because there is no interference evade vector, the interference check evade function of cutter compensation or tool nose radius compensation cannot evade interference.
5447	DANGEROUS AVOIDANCE AT G41/G42	The interference check evade function of cutter compensation or tool nose radius compensation determines that an evade operation will lead to danger.
5448	INTERFERENCE TO AVD. AT G41/G42	In the interference check evade function of cutter compensation or tool nose radius compensation, a further interference occurs for an already created interference evade vector.
5456	TOO MANY G68.2 NESTING	Tilted working plane command G68.2 was specified more than once. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.
5457	G68.2 FORMAT ERROR	A G68.2 format error occurred.
5458	ILLEGAL USE OF G53.1	G53.1 was specified before the G68.2 command.

Number	Message	Description
5459	MACHINE PARAMETER INCORRECT	- A machine configuration parameter (parameter No. 19665 to No.19667 or 19680 to 19714 or No.12321) is illegal. - The axis which is specified in parameter No.19681 or No.19686 is not a rotation axis. - The basic three axes are not specified in the parameter No.1022. - In tool center point control (type 2) or cutter compensation for 5-axis machining (type 2) or tilted working plane command, the end point of rotation axis can not exist within the region specified by parameters No.19741 to No.19744. - In tool center point control (type 2) or cutter compensation for 5-axis machining (type 2), the end point of rotation axis can not exist. Check the machine configuration and the program. - Tool center point control (type 2) or cutter compensation for 5-axis machining (type 2) is specified when hypothetical axis is used. - Tool center point control (type 2) or cutter compensation for 5-axis machining (type 2) is specified when the programming coordinate system is the workpiece coordinate system.

Number	Message	Description
5460	ILLEGAL USE OF TRC FOR 5-AXIS MACHINE	- In the cutter compensation mode for 5-axis machining (except the tool side offset function for a tool rotation type machine), a move command other than G00/G01 is specified. - With a table rotation type machine, when bit 1 (PTD) of parameter No. 19746 is set to 1, a plane selection is made with an axis other than the basic three axes at the start of cutter compensation for 5-axis machining. - When bit 1 (SPG) of parameter No. 19607 is set to 1, there is a discrepancy between the machine type set in parameter No. 19680 and a G code specifying cutter compensation for 5-axis machining (G41.2, G42.2, G41.4, G42.4, G41.5, or G42.5). - With a machine that is not of the tool rotation type, G41.3 is specified. - When bit 5 (WKP) of parameter No. 19696 is set to 0, and bit 4 (TBP) of parameter No. 19746 is set to 0, cutter compensation for 5-axis machining and tool center point control for 5-axis machining are used at the same time. - A rotation axis command is specified in the cutter compensation (type 2) mode for 5-axis machining. - With a table rotation type or mixed type machine, IJK is specified in a block that specifies cutter compensation (type 2) for 5-axis machining (G41.6/G42.6). - An illegal G code is specified in the cutter compensation mode for 5-axis machining. - When cutter compensation for 5-axis machining is specified, the modal state is illegal. - When the table coordinate system is set as the programming coordinate system, table rotation then cutter compensation for 5-axis machining are specified after the start of tool center point control for 5-axis machining. - There is a difference in type1/type 2 specification between cutter compensation for 5-axis machining and tool center point control for 5-axis machining. - When cutter compensation for 5-axis machining and tool center point control for 5-axis machining are used at the same time, one of these functions that is specified earlier than the other is canceled earlier.
5461	ILLEGAL USE OF G41.2/G42.2/G41.5/G42.5	A move command other than G00 or G01 was performed during cutter compensation for 5-axis machining in a mixed-type machine.
5463	ILLEGAL PARAMETER IN TRC FOR 5-AXIS MACHINE	A parameter related to cutter compensation for 5-axis machining is illegal. - Acceleration/deceleration before interpolation is disabled. Set parameter No. 1660. - Rapid traverse acceleration/deceleration before interpolation is disabled. Set bit 1 (LRP) of parameter No. 1401, bit 5 (FRP) of parameter No. 19501, and parameter Nos. No.1671 and 1672.

(4) Parameter writing alarm (SW alarm)

Number	Message	Description
SW0100	PARAMETER ENABLE SWITCH ON	The parameter setting is enabled (PWE, one bit of parameter No. 8000 is set to “1”). To set the parameter, turn this parameter ON. Otherwise, set to OFF.

(5) Servo alarms (SV alarm)

Number	Message	Description
SV0001	SYNC ALIGNMENT ERROR	In feed axis synchronization control, the amount of compensation for synchronization exceeded the parameter (No. 8325) setting value. This alarm occurs only for a slave axis.
SV0002	SYNC EXCESS ERROR ALARM 2	In feed axis synchronization control, the amount of synchronization error exceeded the parameter (No. 8332) setting value. When the synchronization is not completed after power-up, the determination is made by the parameter value (No. 8332) multiplied by the parameter (No. 8330) multiplier. This alarm occurs only for a slave axis only.
SV0003	SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL MODE CAN'T BE CONTINUED	Since as axis in synchronization, composition, or superposition mode caused a servo alarm, the mode could not be continued, If one of the axes in a mode causes a servo alarm, all axes relating to the axis enter the servo-off state. This alarm is generated to enable the cause of the servo-off state to be checked.
SV0004	EXCESS ERROR (G31)	The amount of positional deviation during torque limit skip command operation exceeded the limit value of the parameter No.6287.
SV0005	SYNC EXCESS ERROR (MCN)	In feed axis synchronization control, for synchronization, the difference value of the machine coordinate between a master and slave axes exceeded the parameter (No. 8314) setting value. This alarm occurs for a master or slave axis.
SV0301	APC ALARM: COMMUNICATION ERROR	Since the absolute-position detector caused a communication error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0302	APC ALARM: OVER TIME ERROR	Since the absolute-position detector caused an overtime error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0303	APC ALARM: FRAMING ERROR	Since the absolute-position detector caused a framing error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0304	APC ALARM: PARITY ERROR	Since the absolute-position detector caused a parity error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0305	APC ALARM: PULSE ERROR	Since the absolute-position detector caused a pulse error, the correct machine position could not be obtained. The absolute-position detector, or cable is thought to be defective.
SV0306	APC ALARM: OVER FLOW ERROR	Since the amount of positional deviation overflowed, the correct machine position could not be obtained. Check to see the parameter No. 2084 or No. 2085.

Number	Message	Description
SV0307	APC ALARM: MOVEMENT EXCESS ERROR	Since the machine moved excessively, the correct machine position could not be obtained.
SV0360	ABNORMAL CHECKSUM(INT)	The checksum alarm occurred on the built–in Pulsecoder.
SV0361	ABNORMAL PHASE DATA(INT)	The phase data abnormal alarm occurred on the built–in Pulsecoder.
SV0362	ABNORMAL REV. DATA(INT)	The speed count abnormal alarm occurred on the built-in Pulsecoder.
SV0363	ABNORMAL CLOCK(INT)	The clock alarm occurred on the built–in Pulsecoder.
SV0364	SOFT PHASE ALARM(INT)	A digital servo soft detected an abnormality on the built in Pulsecoder.
SV0365	BROKEN LED(INT)	The digital servo software detected abnormal data on the built–in Pulsecoder.
SV0366	PULSE MISS(INT)	A pulse error occurred on the built–in Pulsecoder.
SV0367	COUNT MISS(INT)	A count error occurred on the built–in Pulsecoder.
SV0368	SERIAL DATA ERROR(INT)	The communications data could not be received from the built–in Pulsecoder.
SV0369	DATA TRANS. ERROR(INT)	A CRC error or stop bit error occurred in the communications data from the built–in Pulsecoder.
SV0380	BROKEN LED(EXT)	Separate detector error
SV0381	ABNORMAL PHASE (EXT)	An abnormal alarm in the position data occurred on the separate linear scale.
SV0382	COUNT MISS(EXT)	A count error occurred on the separate detector.
SV0383	PULSE MISS(EXT)	A pulse error occurred on the separate detector.
SV0384	SOFT PHASE ALARM(EXT)	The digital servo software detected abnormal data on the separate detector.
SV0385	SERIAL DATA ERROR(EXT)	The communications data could not be received from the separate detector.
SV0386	DATA TRANS. ERROR(EXT)	A CRC error or stop bit error occurred in the communications data from the standalone detector.
SV0387	ABNORMAL ENCODER(EXT)	An abnormality occurred on a separate detector. For more information, contact the scale manufacturer.
SV0401	IMPROPER V_READY OFF	Although the ready signal (PRDY) of the position control was ON, the ready signal (VRDY) of the velocity control was OFF.
SV0404	IMPROPER V_READY ON	Although the ready signal (PRDY) of the position control was OFF, the ready signal (VRDY) of the velocity control was ON.
SV0407	EXCESS ERROR	The difference value of the amount of positional deviation for the synchronization axis exceeded the setting value. (during synchronization control only)
SV0409	DETECT ABNORMAL TORQUE	An abnormal load was detected on the servo motor, or during Cs axis or spindle positioning. The alarm can be canceled by RESET.
SV0410	EXCESS ERROR (STOP)	The amount of positional deviation during stopping exceeded the parameter (No. 1829) setting value. In a dual check safety function, an alarm occurs during safety monitoring (when the safety monitoring start signal SEV or SEP is 1), but the alarm cannot be canceled by a reset.
SV0411	EXCESS ERROR (MOVING)	The amount of positional deviation during traveling became excessive than the parameter setting value. (Generally, in the parameter No.1828, the dual check safety function during safety monitoring (when he safety monitoring start signal SEV or SEP is 1) is the parameter No. 1838.) In a dual check safety function, an alarm occurs during safety monitoring, but the alarm cannot be canceled by a reset.
SV0413	LSI OVERFLOW	The counter for the amount of positional deviation overflowed

Number	Message	Description
SV0415	MOTION VALUE OVERFLOW	The velocity exceeding the travel velocity limit was commanded.
SV0417	ILL DGTL SERVO PARAMETER	A digital serve parameter setting is incorrect.
SV0420	SYNC TORQUE EXCESS	In feed axis synchronization control, for synchronization, the difference value of torque between a master and slave axes exceeded the parameter (No. 2031) setting value. This alarm occurs for a master axis.
SV0421	EXCESS ERROR(SEMI-FULL)	The difference between the feedback from the semi and full sides exceeded the setting of parameter No.1729.
SV0422	EXCESS VELOCITY IN TORQUE	In torque control, the commanded permissible velocity was exceeded.
SV0423	EXCESS ERROR IN TORQUE	In torque control, the total permissible move value specified as a parameter was exceeded.
SV0430	SV MOTOR OVERHEAT	The servo motor has overheated.
SV0431	CNV. OVERLOAD	PSM : Overheat β series SVU : Overheat
SV0432	CNV. LOW VOLT CONTROL	PSM : The control power supply voltage has dropped. PSMR : The control power supply voltage has dropped. β series SVU : The control power supply voltage has dropped.
SV0433	CNV. LOW VOLT DC LINK	PSM : Low DC link voltage PSMR : Low DC link voltage α series SVU : Low DC link voltage β series SVU : Low DC link voltage
SV0434	INV. LOW VOLT CONTROL	SVM : Low control power voltage
SV0435	INV. LOW VOLT DC LINK	SVM : Low DC link voltage
SV0436	SOFTTHERMAL(OVC)	The digital servo software detected a software thermal (OVC).
SV0437	CNV. OVERCURRENT POWER	PSM : Overcurrent on input circuit section.
SV0438	INV. ABNORMAL CURRENT	SVM : Motor overcurrent α series SVU : Motor overcurrent β series SVU : Motor overcurrent
SV0439	CNV. OVER VOLT DC LINK	PSM : The DC link voltage is too high. PSMR : The DC link voltage is too high. β series SVU : The DC link voltage is too high.
SV0440	CNV. EX DECELERATION POW.	PSMR : Excessive generative discharge α series SVU : Excessive generative discharge, or abnormal error in generative power circuit
SV0441	ABNORMAL CURRENT OFFSET	The digital servo software detected an abnormality in the motor current detection circuit.
SV0442	CNV. CHARGE FAILURE	PSM : The spare charge circuit for the DC link is abnormal. PSMR : The spare charge circuit for the DC link is abnormal.
SV0443	CNV. COOLING FAN FAILURE	PSM : Internal cooling fan failure. PSMR : Internal cooling fan failure. β series SVU : Internal cooling fan failure.
SV0444	INV. COOLING FAN FAILURE	SVM : Internal cooling fan failure.
SV0445	SOFT DISCONNECT ALARM	The digital servo software detected a disconnected Pulsecoder.
SV0446	HARD DISCONNECT ALARM	The hardware detected a disconnected built–in Pulsecoder.
SV0447	HARD DISCONNECT(EXT)	The hardware detected a disconnected separate detector.
SV0448	UNMATCHED FEEDBACK ALARM	The sign of the feedback signal from the standalone detector is opposite to that from the feedback signal from the built–on Pulsecoder.

Number	Message	Description
SV0449	INV. IPM ALARM	SVM : The IPM (Intelligent Power Module) detected an alarm. α series SVU : The IPM (Intelligent Power Module) detected an alarm.
SV0453	SPC SOFT DISCONNECT ALARM	Software disconnection alarm of the α Pulsecoder. Turn off the power to the CNC, then remove and insert the Pulsecoder cable. If this alarm is issued again, replace the Pulsecoder.
SV0454	ILLEGAL ROTOR POS DETECT	磁極検出機能が異常終了しました。 モータが動かず、磁極位置検出が出来ませんでした。
SV0456	ILLEGAL CURRENT LOOP	An attempt was made to set the current loop that could not be set. The amplifier pulse module in use does not comply with HIGH SPEED HRV. Or, requirements to control are not satisfied in the system.
SV0458	CURRENT LOOP ERROR	The specified current loop differs from the actual current loop.
SV0459	HI HRV SETTING ERROR	For two axes whose servo axis numbers (parameter No. 1023) are consecutively even and odd numbers, HIGH SPEED HRV control is possible for one axis and impossible for the other.
SV0460	FSSB DISCONNECT	The FSSB connection was discontinued. Probable causes are: 1. The FSSB connection cable was disconnected or broken. 2. The amplifier was turned off . 3. In the amplifier, the low-voltage alarm occurred.
SV0462	SEND CNC DATA FAILED	The correct data could not be received on a slave side because of the FSSB communication error.
SV0463	SEND SLAVE DATA FAILED	The correct data could not be received in the servo software because of the FSSB communication error.
SV0465	READ ID DATA FAILED	A read of the ID information for the amplifier has failed at power-on.
SV0466	MOTOR/AMP. COMBINATION	The maximum current of an amplifier is different to that of a motor. Probable causes are: 1. The connection command for an amplifier is incorrect. 2. The parameter (No.2165) setting is incorrect
SV0468	HI HRV SETTING ERROR(AMP)	An attempt was made to set up HIGH SPEED HRV control for use when the controlled axis of an amplifier for which HIGH SPEED HRV control could not be used.
SV0600	INV. DC LINK OVER CURRENT	SVM : DC link overcurrent. β SVU : DC link overcurrent.
SV0601	INV. RADIATOR FAN FAILURE	SVM : Radiator cooling fan failure. β SVU : Radiator cooling fan failure.
SV0602	INV. OVERHEAT	SVM : The servo motor has overheated.
SV0603	INV. IPM ALARM(OH)	SVM : The IPM (Intelligent Power Module) detected an overheat alarm. β SVU : The IPM (Intelligent Power Module) detected an overheat alarm.
SV0604	AMP. COMMUNICATION ERROR	The communication between SVM and PSM is in error.
SV0605	CNV. EX. DISCHARGE POW.	PSMR : The motor regenerative power is too much.
SV0606	CNV. RADIATOR FAN FAILURE	PSM : External radiator cooling fan failure. PSMR : External radiator cooling fan failure.

Number	Message	Description
SV0607	CNV. SINGLE PHASE FAILURE	PSM : The input power supply has a missing phase. PSMR : The input power supply has a missing phase.
SV1025	V_READY ON (INITIALIZING)	The ready signal (VRDY) of the velocity control which should be OFF is ON while the servo control is ON.
SV1026	ILLEGAL AXIS ARRANGE	The parameter for servo axis arrange is not set correctly. A negative value, duplicate value, or greater value than the number of control axes was set to the parameter No. 1023 "The servo axis number of each axis."
SV1055	ILLEGAL TANDEM AXIS	In tandem control, the setting of the parameter No. 1023 is incorrect.
SV1056	ILLEGAL TANDEM PAIR	In tandem control, the setting of the parameter No. 1020, No. 1025, No. 1026 or TDM (No.1817#6) is incorrect.
SV1067	FSSB:CONFIGURATION ERROR(SOFT)	An FSSB configuration error occurred (detected by software). The connected amplifier type is incompatible with the FSSB setting value.
SV1100	S-COMP. VALUE OVERFLOW	The amount of compensation for the straightness exceeded a maximum value of 32767.
SV5134	FSSB:OPEN READY TIME OUT	In the initialization, the FSSB could not be in an open ready sate. The axis card is thought to be defective.
SV5136	FSSB:NUMBER OF AMP. IS INSUFFICIENT	The number of amplifier identified by the FSSB is insufficient than the number of control axes. Or, the setting of the number of axes or the amplifier connection is in error.
SV5137	FSSB:CONFIGURATION ERROR	An FSSB configuration error occurred. The connecting amplifier type is incompatible with the FSSB setting value.
SV5139	FSSB:ERROR	Servo initialization has not completed successfully. It is probable that an optical cable failed or a connection between the amplifier and another module failed.
SV5197	FSSB:OPEN TIME OUT	The initialization of the FSSB was completed, but it could not be opened. Or, the connection between the CNC and the amplifier in is incorrect.
SV5197	FSSB:OPEN TIME OUT	The FSSB could not be opened although the CNC permitted the opening of the FSSB. Check the connection between the CNC and the amplifier.
SV5311	FSSB:ILLEGAL CONNECTION	1. This alarm is issued if axes, whose servo axis numbers (parameter No. 1023) are even and odd numbers, are allocated to the amplifiers connected to the FSSBs of different paths. 2. This alarm is issued if an attempt is made to set up for use of the pulse modules connected to the FSSBs of different paths. And the system did not satisfy the requirements for performing HIGH SPEED HRV control.

(6) Overtravel alarms (OT alarm)

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	Exceeded the positive side stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	Exceeded the negative side stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	Exceeded the positive side stored stroke check 2. Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the positive direction.
OT0503	- OVERTRAVEL (SOFT 2)	Exceeded the negative side stored stroke check 2. Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the negative direction.
OT0504	+ OVERTRAVEL (SOFT 3)	Exceeded the positive side stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	Exceeded the - side stored stroke check 3.
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0508	INTERFERENCE:+	A tool moving in the positive direction along the n axis has fouled another tool post.
OT0509	INTERFERENCE:-	A tool moving in the negative direction along the n axis has fouled another tool post.
OT0510	+ OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the negative direction during the stroke check before movement.
OT0511	- OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the positive direction during the stroke check before movement.
OT1710	ILLEGAL ACC. PARAMETER (OPTIMUM TORQUE ACC/DEC)	The permissible acceleration parameter for the optimum torque acceleration/deceleration is in error. A possible cause is either of the following: (1) The ratio of a negative acceleration to a positive acceleration is not more than the limit value. (2) The time to reduce to a velocity of 0 exceeded the maximum time.

(7) Memory file alarms (IO alarm)

Number	Message	Description
IO1001	FILE ACCESS ERROR	The resident–type file system could not be accessed as an error occurred in the resident–type file system.
IO1002	FILE SYSTEM ERROR	The file could not be accessed as an error occurred in the CNC file system.
IO1030	CHECK SUM ERROR	The checksum of the CNC part program storage memory is incorrect.
IO1032	MEMORY ACCESS OVER RANGE	Accessing of data occurred outside the CNC part program storage memory range.

(8) Alarms requiring power to be turned off (PW alarm)

Number	Message	Description
PW0000	POWER MUST BE OFF	A parameter was set for which the power must be turned OFF then ON again.
PW0001	X-ADDRESS(*DEC) IS NOT ASSIGNED.	The X address of the PMC could not be assigned correctly. This alarm may occur in the following case: - During the setting of parameter No. 3013, the X address could not be assigned correctly for the deceleration dog (*DEC) for a return to the reference position.
PW0002	PMC address is not correct(AXIS).	The address to assign the axis signal is incorrect. This alarm may occur in the following case: - The parameter No.3021 setting is incorrect.
PW0003	PMC address is not correct(SPINDLE).	The address to assign the spindle signal is incorrect. This alarm may occur in the following case: - The parameter No.3022 setting is incorrect.
PW0004	SETTING THE LOADER SYSTEM PATH IS NOT CORRECT.	The loader system could not be assigned correctly. The parameter No. 984 setting is incorrect. - The number of loader systems and the number of systems specified to the loader system in the parameter No. 984#0(LCP) does not match. - The parameter No. 984#0 of the system 1 is set to 1.
PW0006	POWER MUST BE OFF (ILL-EXEC-CHK)	The malfunction prevention function detected an alarm to require the power off.
PW0007	X-ADDRESS(SKIP) IS NOT ASSIGNED	The X address of PMC could not be assigned correctly. Possible causes are: - During the set of parameter No. 3012, the skip signal of the X address was not assigned correctly. - During the set of parameter No. 3019, the address other than the skip signal of the X address was not assigned correctly.
PW1102	ILLEGAL PARAMETER (I-COMP.)	The parameter for setting slope compensation is incorrect. This alarm occurs in the following cases: - When the number of pitch error compensation points on the axis on which slope compensation is executed exceeds 128 between the most negative side and most positive side - When the size relationship between the slope compensation point Nos. is incorrect - When the slope compensation point is not located between the most negative side and most positive side of pitch error compensation - When the compensation per compensation point is too small or too great.
PW1103	ILLEGAL PARAMETER (S-COMP.128)	The parameter for setting 128 straightness compensation points or the parameter compensation data is incorrect,
PW5046	ILLEGAL PARAMETER (S-COMP.)	The parameter for setting straightness compensation is incorrect.

(9) Spindle alarms (SP alarm)

Number	Message	Description
SP0740	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the stopped spindle has exceeded the set value during rigid tapping.
SP0741	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the moving spindle has exceeded the set value during rigid tapping.
SP0742	RIGID TAP ALARM : LSI OVERFLOW	An LSI overflow has occurred for the spindle during rigid tapping.
SP0752	SPINDLE MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contour control, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
SP0754	ABNORMAL TORQUE	An abnormal load was detected in a spindle motor. The alarm can be canceled by RESET.
SP0755	SAFETY FUNCTION ERROR	The CNC CPU detected that the safely function of the n-th spindle was not executed.
SP0756	ILLEGAL AXIS DATA	The CNC CPU detected that the connection state and the hardware setting of the spindle amplifier were incompatible on the n-th spindle. If an alarm occurs because of the configuration change of the spindle amplifier , set the spindle amplifier correctly.
SP0757	SAFETY SPEED OVER	The CNC CPU detected that during safety monitoring (the safety monitoring start signal SEV or SEP is 0), the spindle motor speed was greater than the safety speed (parameter No. 4372, 4438, 4440, or 4442) on the n-th spindle. Operate within the safety speed.
SP1202	SPINDLE SELECT ERROR	In a multi spindle control, the spindle number other than the valid spindle number was selected by a position coder select signal. An attempt was made to select the spindle number of the system having no valid spindle.
SP1210	TOOL CHANGE SP MOTION OVERFLOW	The amount of distribution to a spindle is too much. (specific to the FANUC ROBODRILL)
SP1211	TOOL CHANGE SP ORTN EXCESS ERROR	During a tool change, a too much orientation error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1212	TOOL CHANGE SP MOVE EXCESS ERROR	During a tool change, a too much moving error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1213	TOOL CHANGE SP STOP EXCESS ERROR	During a tool change, a too much stop error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1214	TOOL CHANGE SP ILLEGAL SEQUENCE	During changing tools, an abnormal spindle sequence was detected. (specific to the FANUC ROBODRILL)
SP1220	NO SPINDLE AMP.	Either the cable connected to a serial spindle amplifier is broken, or the serial spindle amplifier is not connected.
SP1221	ILLEGAL MOTOR NUMBER	The spindle No. and the motor No. are incorrectly matched.
SP1224	ILLEGAL SPINDLE-POSITION CODER GEAR RATIO	The spindle–position coder gear ratio was incorrect.
SP1225	CRC ERROR (SERIAL SPINDLE)	A CRC error (communications error) occurred in communications between the CNC and the serial spindle amplifier.

Number	Message	Description
SP1226	FRAMING ERROR (SERIAL SPINDLE)	A framing error occurred in communications between the CNC and the serial spindle amplifier.
SP1227	RECEIVING ERROR (SERIAL SPINDLE)	A receive error occurred in communications between the CNC and the serial spindle amplifier.
SP1228	COMMUNICATION ERROR (SERIAL SPINDLE)	A communications error occurred between the CNC and the serial spindle amplifier.
SP1229	COMMUNICATION ERROR SERIAL SPINDLE AMP.	A communications error occurred between serial spindle amplifiers (motor Nos. 1 and 2, or motor Nos. 3–4).
SP1231	SPINDLE EXCESS ERROR (MOVING)	The position deviation during spindle rotation was greater than the value set in parameters.
SP1232	SPINDLE EXCESS ERROR (STOP)	The position deviation during spindle stop was greater than the value set in parameters.
SP1233	POSITION CODER OVERFLOW	The error counter/speed instruction value of the position coder overflowed.
SP1234	GRID SHIFT OVERFLOW	Grid shift overflowed.
SP1240	DISCONNECT POSITION CODER	The analog spindle position coder is broken.
SP1241	D/A CONVERTER ERROR	The D/A converter for controlling analog spindles is erroneous.
SP1243	ILLEGAL SPINDLE PARAMETER SETTING(GAIN)	The setting for the spindle position gain is incorrect.
SP1244	MOTION VALUE OVERFLOW	The amount of distribution to a spindle is too much
SP1245	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1246	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1247	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1969	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1970	SPINDLE CONTROL ERROR	Initialization of spindle control ended in error.
SP1971	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1972	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1974	ANALOG SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1975	ANALOG SPINDLE CONTROL ERROR	An position coder error was detected on the analog spindle.
SP1976	SERIAL SPINDLE COMMUNICATION ERROR	The amplifier No. could not be set to the serial spindle amplifier.
SP1977	SERIAL SPINDLE COMMUNICATION ERROR	An error occurred in the spindle control software.
SP1978	SERIAL SPINDLE COMMUNICATION ERROR	A time–out was detected during communications with the serial spindle amplifier.
SP1979	SERIAL SPINDLE COMMUNICATION ERROR	The communications sequence was no longer correct during communications with the serial spindle amplifier.
SP1980	SERIAL SPINDLE AMP. ERROR	Defective SIC–LSI on serial spindle amplifier
SP1981	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the analog spindle amplifier side.
SP1982	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the serial spindle amplifier side.
SP1983	SERIAL SPINDLE AMP. ERROR	Could not clear on the spindle amplifier side.
SP1984	SERIAL SPINDLE AMP. ERROR	An error occurred during re–initialization of the spindle amplifier.
SP1985	SERIAL SPINDLE CONTROL ERROR	Failed to automatically set parameters
SP1986	SERIAL SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1987	SERIAL SPINDLE CONTROL ERROR	Defective SIC–LSI on the CNC
SP1988	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1989	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.

Number	Message	Description
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Check to see the following parameter. (No.3716 or 3717)
SP1998	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1999	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.

(10) alarm list (serial spindle)

When a serial spindle alarm occurs, the following number is displayed on the CNC.

NOTE

* Note that the meanings of the SPM indications differ depending on which LED, the red or yellow LED, is on. When the red LED is on, the SPM indicates a 2-digit alarm number. When the yellow LED is on, the SPM indicates an error number that designates a sequence problem (for example, when a rotation command is entered with the emergency stop state not released).
  See "Error Codes (Serial Spindle)."

Number	Message	SPM indication (*1)	Faulty location and remedy	Description
SP9001	SSPA:01 MOTOR OVERHEAT	01	1 Check and correct the peripheral temperature and load status. 2 If the cooling fan stops, replace it.	The thermostat embedded in the motor winding operated. The internal temperature of the motor exceeds the specified level. The motor is used in excess of the continuous rating, or the cooling component is abnormal.
SP9002	SSPA:02 EX DEVIATION SPEED	02	1 Check and correct the cutting conditions to decrease the load. 2 Correct parameter No. 4082.	The motor speed cannot follow a specified speed. An excessive motor load torque is detected. The acceleration/deceleration time in parameter No. 4082 is insufficient.
SP9003	SSPA:03 DC-LINK FUSE IS BROKEN	03	1 Replace the SPM unit. 2 Check the motor insulation status. 3 Replace the interface cable.	The PSM becomes ready (00 is indicated), but the DC link voltage is too low in the SPM. The fuse in the DC link section in the SPM is blown. (The power device is damaged or the motor is ground-fault.) The JX1A/JX1B connection cable is abnormal.
SP9004	SSPA:04 POWER SUPPLY ERROR	04	Check the state of the input power supply to the PSM.	The PSM found a missing power supply phase. (PSM alarm 5)
SP9006	THERMAL SENSOR DISCONNECT	06	1 Check and correct the parameter. 2 Replace the feedback cable.	The temperature sensor of the motor is disconnected.
SP9007	SSPA:07 OVER SPEED	07	Check for a sequence error. (For example, check whether spindle synchronization was specified when the spindle could not be turned.)	The motor speed has exceeded 115% of its rated speed. When the spindle axis was in position control mode, positional deviations were accumulated excessively (SFR and SRV were turned off during spindle synchronization.)

Number	Message	SPM indication (*1)	Faulty location and remedy	Description
SP9009	SSPA:09 OVERHEAT MAIN CIRCUIT	09	1 Improve the heat sink cooling status. 2 If the heat sink cooling fan stops, replace the SPM unit.	Abnormal temperature rise of the power transistor radiator
SP9011	SSPA:11 OVERVOLT POWER CIRCUIT	11	1 Check the selected PSM. 2 Check the input power voltage and change in power during motor deceleration. If the voltage exceeds 253 VAC (for the 200-V system) or 530 VAC (for the 400-V system), improve the power supply impedance.	Overvoltage of the DC link section of the PSM was detected. (PSM alarm indication: 7) PSM selection error. (The maximum output specification of the PSM is exceeded.)
SP9012	SSPA:12 OVERCURRENT POWER CIRCUIT	12	1 Check the motor insulation status. 2 Check the spindle parameters. 3 Replace the SPM unit.	The motor output current is abnormally high. A motor-specific parameter does not match the motor model. Poor motor insulation
SP9013	SSPA:13 CPU DATA MEMORY FAULT	13	Replace the SPM control printed circuit board.	Abnormality in an SPM control circuit component is detected. (RAM within the SPM is abnormal.)
SP9015	SSPA:15 SPINDLE SWITCHING FAULT	15	1 Check and correct the ladder sequence. 2 Replace the switching MC.	The switch sequence in spindle switch/output switch operation is abnormal. The switching MC contact status check signal and command do not match.
SP9016	SSPA:16 RAM ERROR	16	Replace the SPM control printed circuit board.	Abnormality in an SPM control circuit component is detected. (RAM for external data is abnormal.)
SP9018	SSPA:18 SUMCHECK ERROR PROGRAM ROM	18	Replace the SPM control printed circuit board.	Abnormality in an SPM control circuit component is detected. (Program ROM data is abnormal.)
SP9019	SSPA:19 EXCESS OFFSET CURRENT U	19	Replace the SPM unit.	Abnormality in an SPM component is detected. (The initial value for the U phase current detection circuit is abnormal.)
SP9020	SSPA:20 EXCESS OFFSET CURRENT V	20	Replace the SPM unit.	Abnormality in an SPM component is detected. (The initial value of the V phase current detection circuit is abnormal.)
SP9021	POS SENSOR POLARITY ERROR	21	Check and correct the parameters. (No. 4000#0, 4001#4)	The polarity parameter setting of the position sensor is wrong.
SP9024	SSPA:24 SERIAL TRANSFER ERROR	24	1 Place the CNC-to-spindle cable away from the power cable. 2 Replace the cable.	The CNC power is turned off (normal power-off or broken cable). An error is detected in communication data transferred to the CNC.

Number	Message	SPM indication (*1)	Faulty location and remedy	Description
SP9027	SSPA:27 DISCONNECT POSITION CODER	27	1 Replace the cable. 2 Re-adjust the BZ sensor signal.	1 The spindle position coder (connector JY4) signal is abnormal. 2 The signal amplitude (connector JY2) of the MZ or BZ sensor is abnormal. (Unconnected cable, adjustment error, etc.)
SP9029	SSPA:29 OVERLOAD	29	Check and correct the load status.	Excessive load has been applied continuously for a certain period of time. (This alarm is issued also when the motor shaft has been locked in the excitation state.)
SP9030	SSPA:30 OVERCURRENT INPUT CIRCUIT	30	Check and correct the power supply voltage.	Overcurrent is detected in PSM main circuit input. (PSM alarm indication: 1) Unbalanced power supply. PSM selection error (The maximum PSM output specification is exceeded.)
SP9031	SSPA:31 MOTOR LOCK OR DISCONNECT DETECTOR	31	1 Check and correct the load status. 2 Replace the motor sensor cable (JY2 or JY5).	The motor cannot rotate at a specified speed. (A level not exceeding the SST level for the rotation command has existed continuously.) Abnormality in the speed detection signal.
SP9032	SSPA:32 SIC-LSI RAM FAULT	32	Replace the SPM control printed circuit board.	Abnormality in an SPM control circuit component is detected. (The LSI device for serial transfer is abnormal.)
SP9033	SSPA:33 SHORTAGE POWER CHARGE	33	1 Check and correct the power supply voltage. 2 Replace the PSM unit.	Charging of direct current power supply voltage in the power circuit section is insufficient when the magnetic contractor in the amplifier is turned on (such as open phase and defective charging resistor).
SP9034	SSPA:34 ILLEGAL PARAMETER	34	Correct a parameter value according to the manual. If the parameter number is unknown, connect the spindle check board, and check the indicated parameter.	Parameter data exceeding the allowable limit is set.
SP9036	SSPA:36 OVERFLOW ERROR COUNTER	36	Check whether the position gain value is too large, and correct the value.	An error counter overflow occurred.
SP9037	SSPA:37 ILLEGAL SETTING VELOCITY DETECTOR	37	Correct the value according to the parameter manual.	The setting of the parameter for the number of pulses in the speed detector is incorrect.

Number	Message	SPM indication (*1)	Faulty location and remedy	Description
SP9041	SSPA:41 ILLEGAL 1REV SIGN OF POSITION CODER	41	1 Check and correct the parameter. 2 Replace the cable. 3 Re-adjust the BZ sensor signal. Re-adjust the BZ sensor signal.	1 The 1-rotation signal of the spindle position coder (connector JY4) is abnormal. 2 The 1-rotation signal (connector JY2) of the MZ or BZ sensor is abnormal. 3 Parameter setting error
SP9042	SSPA:42 NO 1REV SIGN OF POSITION CODER	42	1 Replace the cable. 2 Re-adjust the BZ sensor signal.	1 The 1-rotation signal of the spindle position coder (connector JY4) is disconnected. 2 The 1-rotation signal (connector JY2) of the MZ or BZ sensor is disconnected.
SP9043	SSPA:43 DISCONNECT POSITION CODER DEF. SPEED	43	Replace the cable.	The differential speed position coder signal (connector JY8) in SPM type 3 is abnormal.
SP9046	SSPA:46 ILLEGAL 1REV SIGN OF SCREW CUT	46	1 Check and correct the parameter. 2 Replace the cable. 3 Re-adjust the BZ sensor signal.	An abnormality equivalent to alarm 41 was detected during thread cutting operation.
SP9047	SSPA:47 ILLEGAL SIGNAL OF POSITION CODER	47	1 Replace the cable. 2 Re-adjust the BZ sensor signal. 3 Correct the cable layout (vicinity of the power line).	1 The A/B phase signal of the spindle position coder (connector JY4) is abnormal. 2 The A/B phase signal (connector JY2) of the MZ or BZ sensor is abnormal. The relationship between the A/B phase and 1-rotation signal is incorrect (Pulse interval mismatch).
SP9049	SSPA:49 DEF. SPEED IS OVER VALUE	49	Check whether the calculated differential speed value exceeds the maximum motor speed.	In differential speed mode, the speed of the other spindle converted to the speed of the local spindle has exceeded the allowable limit (the differential speed is calculated by multiplying the speed of the other spindle by the gear ratio).
SP9050	SSPA:50 SYNCRONOUS VALUE IS OVER SPEED	50	Check whether the calculated value exceeds the maximum motor speed.	In spindle synchronization, the speed command calculation value exceeded the allowable limit (the motor speed is calculated by multiplying the specified spindle speed by the gear ratio).
SP9051	SSPA:51 LOW VOLT POWER CIRCUIT	51	1 Check and correct the power supply voltage. 2 Replace the MC.	Input voltage drop was detected. (PSM alarm indication: 4) (Momentary power failure or poor MC contact)
SP9052	SSPA:52 ITP FAULT 1	52	1 Replace the SPM control printed circuit board. 2 Replace the spindle interface printed circuit board in the CNC.	NC interface abnormality was detected (the ITP signal stopped).

Number	Message	SPM indication (*1)	Faulty location and remedy	Description
SP9053	SSPA:53 ITP FAULT 2	53	1 Replace the SPM control printed circuit board. 2 Replace the spindle interface printed circuit board in the CNC.	NC interface abnormality was detected (the ITP signal stopped).
SP9054	SSPA:54 OVERCURRENT	54	Review the load state.	An overload current was detected.
SP9055	SSPA:55 ILLEGAL POWER LINE	55	1 Replace the magnetic contactor. 2 Check and correct the sequence.	The power line state signal of the magnetic contactor for selecting a spindle or output is abnormal.
SP9056	COOLING FAN FAILURE	56	Replace the SPM unit.	The cooling fan in the SPM control circuit stopped.
SP9057	CONV. EX. DECELERATION POW.	57	1 Decrease the acceleration/deceleration duty. 2 Check the cooling condition (peripheral temperature). 3 If the cooling fan stops, replace the resistor. 4 If the resistance is abnormal, replace the resistor.	An overload was detected in the regenerative resistance. (PSMR alarm indication: 8) Thermostat operation or short-time overload was detected. The regenerative resistor was disconnected, or an abnormal resistance was detected.
SP9058	CNV. OVERLOAD	58	1 Check the PSM cooling status. 2 Replace the PSM unit.	The temperature of the radiator of the PSM has increased abnormally. (PSM alarm indication: 3)
SP9059	CNV. COOLING FAN FAILURE	59	Replace the PSM unit.	The cooling fan in the PSM stopped. (PSM alarm indication: 2)
SP9061	SSPA:61 DECODED ALARM	61	Check parameter settings.	The error between the semi-closed and full-closed sides when the dual position feedback function is used is too large.
SP9065	SSPA:65 DECODED ALARM	65	1 Check parameter settings. 2 Check sensor connections and signals. 3 Check power line connections.	The move distance is too long when the magnetic pole is confirmed (synchronization spindle)
SP9066	COM. ERROR BETWEEN SP AMPS	66	1 Replace the cable. 2 Check and correct the connection.	An error was found in communication between amplifiers.
SP9069	SAFETY SPEED OVER	69	1 Check the specified speed. 2 Check parameter settings. 3 Check the sequence.	In the state in which safety speed monitoring was enabled, the system detected that the motor speed exceeded the safety speed or detected an error during a free-run stop.
SP9070	ILLEGAL AXIS DATA	70	1 Check connections (JA7A of the second spindle requires a dedicated connector). 2 Replace the SPM control printed-circuit board.	An error was detected in an axis number check.

Number	Message	SPM indication (*1)	Faulty location and remedy	Description
SP9071	SAFETY PARAMETER ERROR	71	Replace the SPM control printed-circuit board.	An error was detected in an axis parameter check.
SP9072	MISMATCH RESULT OF MOTOR SPEED CHECK	72	1 Replace the SPM control printed-circuit board. 2 Replace the spindle interface printed circuit board in the CNC.	A mismatch was detected between the safety speed check results of the SPM and those of the CNC.
SP9073	MOTOR SENSOR DISCONNECTED	73	1 Replace the feedback cable. 2 Check the shield processing. 3 Check and correct the connection. 4 Adjust the sensor.	The motor sensor feedback signal is not present.
SP9074	CPU TEST ERROR	74	Replace the SPM control printed-circuit board.	An error was detected in a CPU test.
SP9076	INEXECUTION OF SAFETY FUNCTIONS	76	Replace the SPM control printed-circuit board.	The SPM detected that safety functions were not executed.
SP9077	MISMATCH RESULT OF AXIS NUMBER CHECK	77	1 Replace the SPM control printed-circuit board. 2 Replace the spindle interface printed circuit board in the CNC.	A mismatch was detected between the axis number check results of the SPM and those of the CNC.
SP9078	MISMATCH RESULT OF SAFETY PARAMETER CHECK	78	1 Replace the SPM control printed-circuit board. 2 Replace the spindle interface printed circuit board in the CNC.	A mismatch was detected between the safety parameter check results of the SPM and those of CNC.
SP9080	ALARM AT THE OTHER SP AMP.	80	Remove the cause of the alarm of the remote SPM.	During inter-SPM communication, an alarm was generated on the remote SPM.
SP9081	1-ROT MOTOR SENSOR ERROR	81	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the motor sensor cannot be correctly detected.
SP9082	NO 1-ROT MOTOR SENSOR	82	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the motor sensor is not generated.
SP9083	MOTOR SENSOR SIGNAL ERROR	83	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a motor sensor feedback signal.
SP9084	SPNDL SENSOR DISCONNECTED	84	1 Replace the feedback cable. 2 Check the shield processing. 3 Check and correct the connection. 4 Check and correct the parameter. 5 Adjust the sensor.	The spindle sensor feedback signal is not present.
SP9085	1-ROT SPNDL SENSOR ERROR	85	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the spindle sensor cannot be correctly detected.
SP9086	NO 1-ROT SPNDL SENSOR	86	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the spindle sensor is not generated.

Number	Message	SPM indication (*1)	Faulty location and remedy	Description
SP9087	SPNDL SENSOR SIGNAL ERROR	87	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a spindle sensor feedback signal.
SP9088	COOLING RADI FAN FAILURE	88	Replace the SPM external cooling fan.	The external cooling fan stopped.
SP9089	SSPA:89 DECODED ALARM	89	1 Check the connection between the SPM and the submodule SM (SSM). 2 Replace the submodule SM(SSM). 3 Replace the SPM control printed-circuit board.	Submodule SM (SSM) error (synchronous spindle)
SP9110	AMP COMMUNICATION ERROR	b0	1 Replace the communication cable between amplifier and module. 2 Replace the SPM or PSM control printed circuit board.	Communication error between amplifier and module
SP9111	CONV. LOW VOLT CONTROL	b1	Replace the PSM control printed circuit board.	Low converter control power supply voltage (PSM indication = 6)
SP9112	CONV. EX. DISCHARGE POW.	b2	1 Check the regenerative resistance. 2 Check the motor selection. 3 Replace the PSM	Excessive converter regenerative power (PSM indication = 8)
SP9113	CONV. COOLING FAN FAILURE	b3	Replace the cooling fan.	Stopped cooling fan of the converter radiator (PSM indication = A)
SP9120	COMMUNICATION DATA ERROR	C0	1 Replace the communication cable between CNC and SPM. 2 Replace the SPM control printed circuit board. 3 Replace the CNC side spindle interface printed circuit board.	Communication data alarm
SP9121	COMMUNICATION DATA ERROR	C1	1 Replace the communication cable between CNC and SPM. 2 Replace the SPM control printed circuit board. 3 Replace the CNC side spindle interface printed circuit board.	Communication data alarm
SP9122	COMMUNICATION DATA ERROR	C2	1 Replace the communication cable between CNC and SPM. 2 Replace the SPM control printed circuit board. 3 Replace the CNC side spindle interface printed circuit board.	Communication data alarm
SP9123	SSPA:C3 DECODED ALARM	C3	Replace the submodule SW(SSW).	Submodule SW (SSW) error (spindle switching)

Error codes (serial spindle)

> **NOTE**
> *1 Note that the meanings of the SPM indications differ depending on which LED, the red or yellow LED, is on. When the yellow LED is on, an error code is indicated with a 2-digit number. An error code is indicated in the CNC diagnosis, No.712. When the red LED is on, the SPM indicates the number of an alarm generated in the serial spindle.
> → See “(10) Serial spindle alarms (SP alarm).”

SPM indication (*1)	Faulty location and remedy	Description
01	Although neither *ESP (emergency stop signal; there are two types of signals including the input signal and PSM contact signal) nor MRDY (machine ready signal) is input, SFR (forward rotation signal)/SRF (reverse rotation signal)/ORCM (orientation command) is input.	Check the *ESP and MRDY sequence. For MRDY, pay attention to the parameter setting regarding the use of the MRDY signal (parameter No. 4001#0).
03	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No. 4002 = 0, 0, 0, 0), but a Cs contour control command is input. In this case, the motor is not excited.	Check the parameter settings.
04	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No.4002 = 0, 0, 0,), but a servo mode (rigid tapping, spindle positioning, etc.) or spindle synchronization command is input. In this case, the motor is not excited.	Check the parameter settings.
05	The orientation function option parameter is not specified, but ORCM (orientation command) is input.	Check the orientation function parameter settings.
06	The output switching control function option parameter is not specified, but low-speed characteristic winding is selected (RCH = 1).	Check the output switching control function parameter settings and the power line state check signal (RCH).
07	A Cs contour control command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
08	A servo mode (rigid tapping, spindle positioning, etc.) control command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
09	A spindle synchronization command is input, but SFR (clockwise rotation command)/SRV (counterclockwise rotation command) is not input.	Check the sequence.
10	A Cs contour control command is input, but another mode (servo mode, spindle synchronization, or orientation) is specified.	Do not switch to another mode during a Cs contour control command. Before moving to another mode, cancel the Cs contour control command.

SPM indication (*1)	Faulty location and remedy	Description
11	A servo mode (rigid tapping, spindle positioning, etc.) command is input, but another mode (Cs contour control, spindle synchronization, or orientation) is specified.	Do not switch to another mode during a servo mode command. Before moving to another mode, cancel the servo mode command.
12	A spindle synchronization command is input, but another mode (Cs contour control, servo mode, or orientation) is specified.	Do not switch to another mode during a spindle synchronization command. Before moving to another mode, cancel the spindle synchronization command.
14	Both SFR (clockwise rotation command) and SRV (counterclockwise rotation command) are input at the same time.	Issue either of them.
17	The speed detector parameter settings (bits 2, 1, and 0 of parameter No. 4011) are not valid. ☆ There is no corresponding speed detector.	Check the parameter settings.
18	The parameter settings are such that a position sensor is not used (position control not performed (bits 3, 2, 1, and 0 of parameter No. 4002), but position coder system orientation is issued.	Check the parameter settings and the input signal.
24	If index is performed continuously in position coder system orientation, an incremental operation is performed first (INCMD = 1), then an absolute position command (INCMD = 0) is input.	Check INCMD (incremental command). If an absolute position command is to follow, be sure to perform absolute position command orientation first.
29	The parameter settings are such that the shortest-time orientation function is used (bit 6 of parameter No. 4018 = 0, Nos. 4320 to 4323 ≠0).	In the αi series spindle amplifier, the shortest-time orientation function cannot be used. Use normal-system orientation.
31	The hardware configuration is such that the spindle FAD function cannot be used. In this case, the motor is not activated.	Check the CNC model.
33	The hardware configuration is such that the spindle EGB function cannot be used. In this case, the motor is not activated.	Check the CNC model.
34	Both the spindle FAD function and the spindle EGB function are enabled. In this case, the motor is not activated.	The two functions cannot be used at the same time. Enable either function only.
34	The submodule SM (SSM) is faulty or the connection between SPM and SSM is in error.	Submodule SM (SSM) error (synchronous spindle)

***2**
*2 PSM contact signal
Between ESP1 and ESP2 on the PSM
Contact open: Emergency stop
Contact closed: Normal operation

(11) Overheat alarms (OH alarm)

Number	Message	Description
OH0700	LOCKER OVERHEAT	CNC cabinet overheat
OH0701	FAN MOTOR STOP	PCB cooling fan motor abnormality

(12) Other alarms (DS alarm)

Number	Message	Description
DS0001	SYNC EXCESS ERROR (POS DEV)	In feed axis synchronization control, the difference in the amount of positional deviation between the master and slave axes exceeded the parameter (No. 8323) setting value. This alarm occurs only for the slave axis.
DS0002	SYNC EXCESS ERROR ALARM 1	In feed axis synchronization control, the difference in the amount of synchronization between the master and slave axes exceeded the parameter (No. 8331) setting value. This alarm occurs only for the slave axis.
DS0003	SYNCHRONIZE ADJUST MODE	The system is in the synchronize adjust mode.
DS0004	EXCESS MAXIMUM FEEDRATE	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
DS0005	EXCESS MAXIMUM ACCELERATION	The malfunction prevention function detected the command in which a value exceeding the maximum acceleration was specified.
DS0014	TOOL CHANGE DETECT MACHINE LOCK	A machine lock is turned on for the Z axis for which the tool is being changed.
DS0015	TOOL CHANGE DETECT MIRROR IMAGE	A mirror image is turned on for the Z axis for which the tool is being changed.
DS0020	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the perpendicular axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the perpendicular axis.
DS0024	MISMATCH OF ANGULAR AXIS(D.C.S)	On angular axis control, one of the angular/perpendicular axes is the scale with ref-pos, and the other of them is not the scale with ref-pos. Such system is not admired.
DS0026	MISMATCH OF ANGULAR AXIS(D.C.S)	On angular axis control, one of the angular/perpendicular axes is the scale with ref-pos, and the other of them is not the scale with ref-pos. Such system is not admired.
DS0027	MISMATCH OF SYNCHRONOUS AXIS(D.C.S)	Master/slave axes of feed axis synchronization control, one of them is the linear scale with distance-coded reference marks, and the other of them is not the linear scale with distance-coded reference marks. Please establish reference position with the input signal SYNCn<G138>, SYNCJn<G140> or pameter setting to 0.
DS0059	SPECIFIED NUMBER NOT FOUND	[External data I/O] The No. specified for a program No. or sequence No. search could not be found. There was an I/O request issued for a pot No. or offset (tool data), but either no tool numbers have been input since power ON or there is no data for the entered tool No. [External workpiece No. search] The program corresponding to the specified workpiece No. could not be found.

Number	Message	Description
DS0131	TOO MANY MESSAGE	An attempt was made to display an external operator message or external alarm message, but five or more displays were required simultaneously.
DS0132	MESSAGE NUMBER NOT FOUND	An attempt to cancel an external operator message or external alarm message failed because the specified message number was not found.
DS0133	TOO LARGE NUMBER	A value other than 0 to 4095 was specified as the external operator message or the external alarm message number.
DS0300	APC ALARM: NEED REF RETURN	A setting to zero position for the absolute position detector (association with reference position and the counter value of the absolute position detector) is required. Perform the return to the reference position. This alarm may occur with other alarms simultaneously. In this case, other alarms must be handled first.
DS0306	APC ALARM: BATTERY VOLTAGE 0	The battery voltage of the absolute position detector has dropped to a level at which data can no longer be held. Or, the power was supplied to the Pulsecoder for the first time. The battery or cable is thought to be defective. Replace the battery with the machine turned on.
DS0307	APC ALARM: BATTERY LOW 1	The battery voltage of the absolute position detector has dropped to a level at which a replacement is required. Replace the battery with the machine turned on.
DS0308	APC ALARM: BATTERY LOW 2	The battery voltage of the absolute position detector dropped to a level at which a replacement was required in the past. (including during power off) Replace the battery with the machine turned on.
DS0309	APC ALARM: REF RETURN IMPOSSIBLE	An attempt was made to set the zero point for the absolute position detector by MDI operation when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
DS0310	NOT ON RETURN POINT	The return position recorded during retraction is not reached during recovery. The position may be displaced during recovery due to a machine lock or mirror image. Perform the operation again after making a reset.
DS0405	ZERO RETURN END NOT ON REF	The axis specified in automatic zero return was not at the correct zero point when positioning was completed. Perform zero return from a point whose distance from the zero return start position to the zero point is 2 or more revolutions of the motor. Other probable causes are: - The positional deviation after triggering the deceleration dog is less than 128. - Insufficient voltage or malfunctioning Pulsecoder.
DS1120	UNASSIGNED ADDRESS (HIGH)	The upper 4 bits (EIA4 to EIA7) of an external data I/O interface address signal are set to an undefined address (high bits).
DS1121	UNASSIGNED ADDRESS (LOW)	The lower 4 bits (EIA0 to EIA3) of an external data I/O interface address signal are set to an undefined address (low bits).
DS1124	OUTPUT REQUEST ERROR	OUTPUT REQUEST ERROR An output request was issued during external data output, or an output request was issued for an address that has no output data.

Number	Message	Description
DS1127	DI.EIDHW OUT OF RANGE	The numerical value input by external data input signals EID32 to EID47 has exceeded the permissible range.
DS1128	DI.EIDLL OUT OF RANGE	The numerical value input by external data input signals EID0 to EID31 has exceeded the permissible range.
DS1130	SEARCH REQUEST NOT ACCEPTED	No requests can be accepted for a program No. or a sequence No. search as the system is not in the memory mode or the reset state.
DS1131	EXT-DATA ERROR (OTHER)	[External Data I/O] An attempt was made to input tool data for tool offset by a tool No. during loading by the G10 code.
DS1150	A/D CONVERT ALARM	A/D converter malfunction
DS1184	PARAMETER ERROR IN TORQUE	An invalid parameter was set for torque control. The torque constant parameter is set to “0”.
DS1185	OVER MAXIMUM FEED	The maximum cutting feedrate or rapid traverse feedrate was exceeded in G54.3.
DS1448	ILLEGAL PARAMETER (D.C.S.)	The setting value of parameter for reference marks is satisfied the following any conditions. - The absolute-position detection function is enabled. - Either parameter 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0. - Parameters 1821 and 1882 have identical settings. - The difference between the settings made for parameters 1821 and 1882 is greater than or equal to twice either setting. - The setting value of parameters 1883 and 1884 are over the valid data range.
DS1449	REFERENCE MARK ARE DIFFERENT FROM PARAMETER	In case of distance coded linear scale I/F, the actual interval of reference marks is different from parameter (No.1821,1882) setting value.
DS1450	ZERO RETURN NOT FINISHED	1st reference position return (CDxX7 to CDxX0: 17h (Hex)) was specified when the manual reference position return was not executed with the reference position return function enabled (parameter ZRN (No. 1005#0) set to “0”).
DS1451	IMPROPER PMC AXIS COMMAND	The PMC axes cannot be controlled in this state.
DS1512	EXCESS VELOCITY	The feedrate of the linear axis during polar coordinate interpolation exceeded the maximum cutting feedrate.
DS1514	ILLEGAL MOTION IN G12.1 MODE	In a hypothetical axis direction compensation during the polar coordinate interpolation mode, an attempt is made to travel to the area in which the travel cannot be made.
DS1553	EXCESS VELOCITY IN G43.4/G43.5	The axis rate was attempt to exceed the maximum cutting feedrate and travel by the pivot tool length compensation.
DS1710	ILLEGAL ACC. PARAMETER (OPTIMUM TORQUE ACC/DEC)	There are errors in the parameters of permissible acceleration for Optimum Torque Acceleration/Deceleration. One of the following is the cause. 1) The ratio of the acceleration for deceleration to the acceleration for the acceleration is lower than the limited value. 2) The time to decelerate to 0 is larger than the maximum.
DS1931	MACHINE PARAMETER INCORRECT	One of parameters Nos. 19665 to 19667 and Nos.19680 to 19744 used to configure the machine contains an error.
DS1932	DI.THML SIGNAL ON	One of the parameters used to configure the machine is rewritten while the tool direction thermal displacement compensation function is enabled.

Number	Message	Description
DS1933	NEED REF RETURN(SYNC:MIX:OVL)	The relation between a machine coordinate of an axis in synchronization, composition, or superposition control, and the absolute, or relative coordinate was displaced. Perform the manual return to the reference position.

(13) Malfunction prevention function alarms (IE alarm)

Number	Message	Description
IE0001	+ OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the positive side was exceeded.
IE0002	- OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the negative side was exceeded.
IE0003	+ OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the positive side was exceeded.
IE0004	- OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the negative side was exceeded.
IE0005	+ OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the positive side was exceeded.
IE0006	- OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the negative side was exceeded.
IE0007	EXCESS MAXIMUM REV. DATA	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
IE0008	ILLEGAL ACC/DEC	The malfunction prevention function detected the acceleration/deceleration error.
IE0009	ILLEGAL MCN COODINATE	The malfunction prevention function detected the displacement of a machine coordinate in the check point.

H PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING

H.1 PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING

Overview

By using this PC tool, you can make the memory card program file ("FANUCPRG.BIN") which is needed for the function "Memory Card Program Operation/Editing".
The maximum size of the memory card program file is 2048 Mbytes (2 Gbytes). The "Memory Card Program Operation/Editing" function needs the memory card which has the memory card program file on the FAT formatted memory card.
This PC tool can be operated on a PC in the marketplace and with following OS :

- Windows(R)NT4.0 Workstation (SP5 or later)
- Windows(R)2000 Professional

And acceptable specification is as followings :

- Memory : 32Mbytes or more
- Hard disk : 10Mbytes or more free space and additional space for the memory card program file

H.1.1 Usage Notes

Before using this PC tool, please make sure there is no [temp] folder on the same place of this PC tool.
[temp] folder is created and used by this PC tool as work folder.
If [temp] folder is created, please do not access this folder.
[temp] folder and the files located in this folder will be deleted by this PC tool.

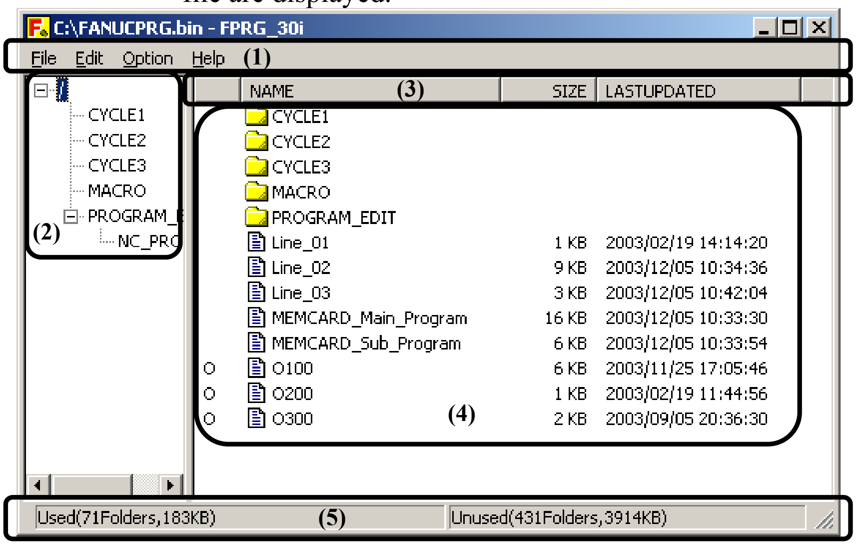
H.1.2 List of Functions of PC Tool

- Browsing the folders of the memory card program file
- Adding a program to the memory card program file by dropping the program into this PC tool from Explorer etc. (hereinafter referred to as "drop in")
- Extracting a program in the memory card program file as a text file on Windows file system by dropping out from this PC tool into Explorer etc. (hereinafter referred to as "drop out")
- Renaming a program in the memory card program file
- Deleting a program in the memory card program file
- Creating a new folder into the memory card program file
- Renaming a folder in the memory card program file
- Deleting a folder in the memory card program file
- Display of free space on the memory card program file
- Sorting list view of the memory card program file

H.1.3 Explanation Of Operations

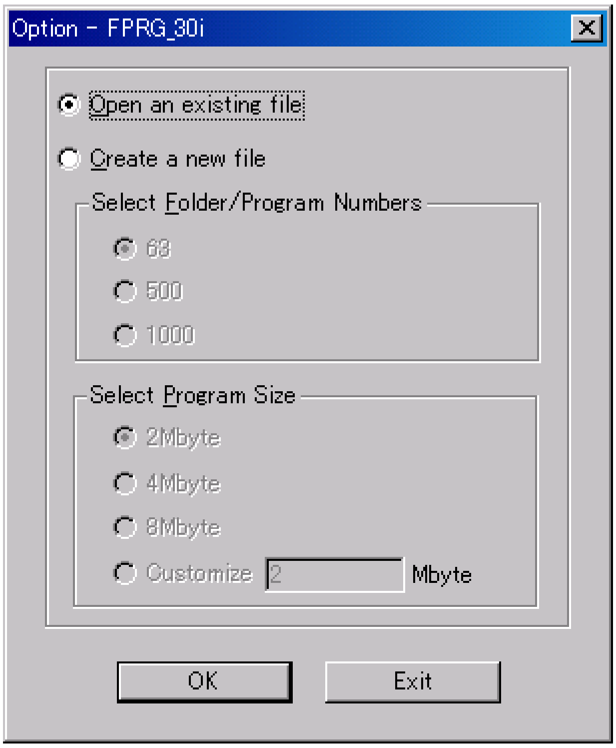
- Outline of screen

1) Menu bar : The menu of this PC tool is displayed.
2) Tree view : Browsing the folders of the memory card program file.
3) Column : Attributes of each file or folder in the memory card program file.
4) List view : Contents of selected folder are displayed.
5) Status bar: Used and Unused space in the memory card program file are displayed.

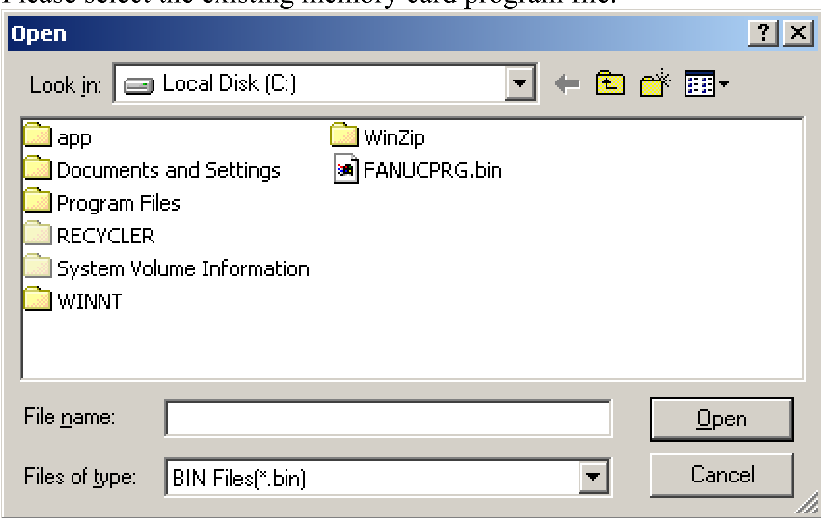


- Initial Option dialogue window

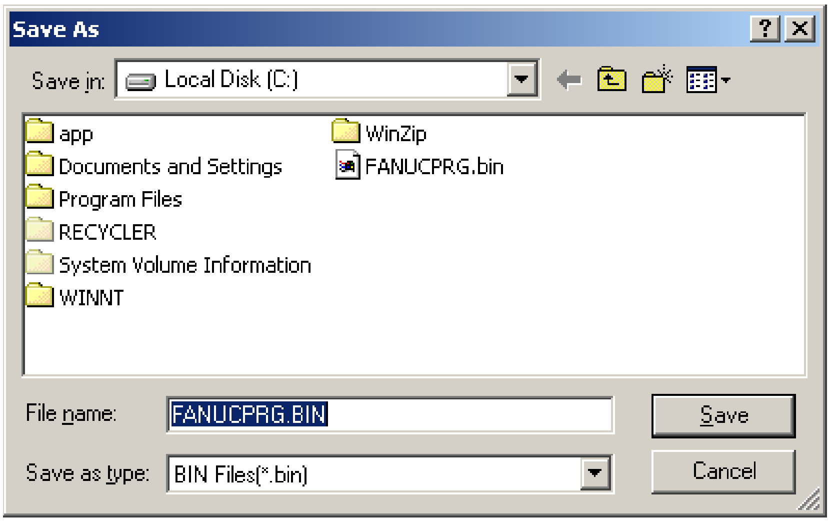
When this PC tool starts up, the Option dialogue window is displayed. Please select "Open an existing file" or "Create a new file".



-When "Open an existing file" is selected
After OK button pushed, "Open" dialogue window is displayed.
Please select the existing memory card program file.



-When "Create a new file" is selected
After OK button pushed, "Save As" dialogue window is displayed.
Please create a new memory card program file on the selected folder.



When the new the memory card program file is created, the following items need to be selected:

- Folder/Program Numbers
- Program Size

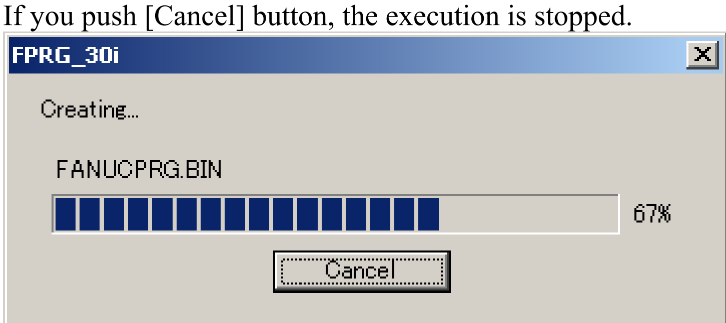
"Folder/Program Numbers" can be selected among 63 / 500 / 1000. The default value is 63.

"Program Size" can be selected among 2Mbyte, 4Mbyte, 8Mbyte, and Customize. The default value is 2Mbyte.

NOTE

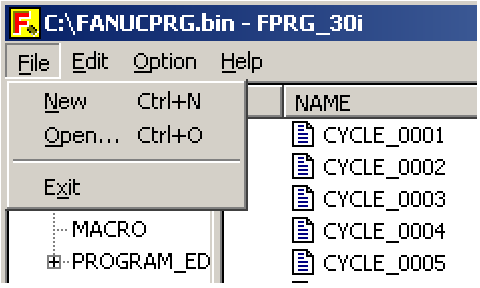
1 When "Customize" size is selected, it is available to cover the range from 2Mbyte to 2048Mbyte.
2 Though the maximum size is 2048Mbyte, a slight loss exists by system use.
3 The number of "Folders" in the status bar includes number of program file and folder.

During creating of the memory card program file, the progress bar is being displayed. This progress bar is also displayed during executing of Drop-in and Drop-out.
If you push [Cancel] button, the execution is stopped.



- Menu

File menu



[New]

Create a new memory card program file.

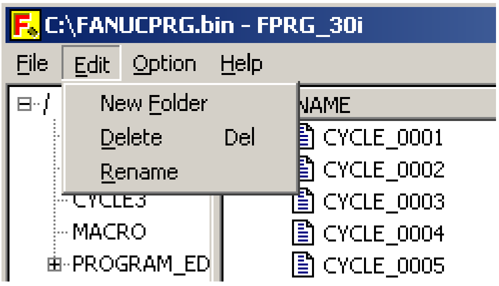
[Open...]

Open the existing memory card program file.

[Exit]

Terminate this PC tool.

Edit menu



[New Folder]

Create new folder. It is available during Tree view selected.

Up to seven hierarchical levels starting from the user root folder (/USER) are permitted.

/USER/PATH1/Aaa/Bbb/Ccc/Ddd/O123

1 2 3 4 5 6 7(not folder)

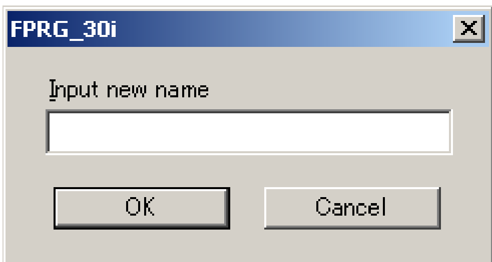
[Delete]

Delete program files or folders.

If you delete a folder, all folders and program files in the folder will be deleted.

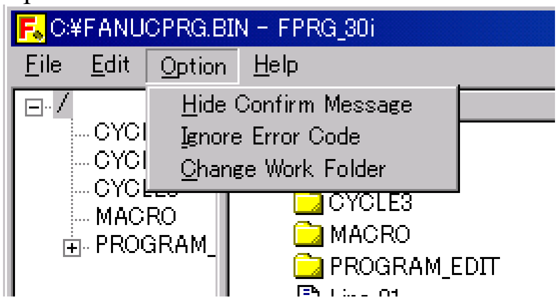
[Rename]

Rename a folder or file.



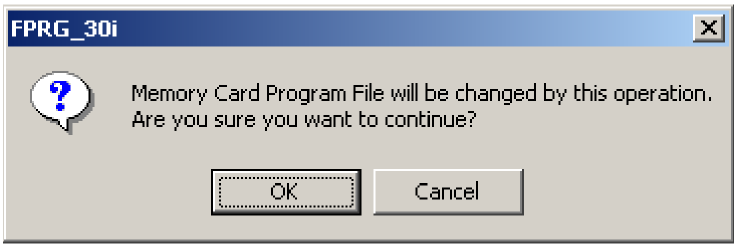
> **NOTE**
> For naming folder and program file, characters which can be used are limited. Please refer to "Naming rules".

Option menu



[Hide Confirm Message]

When the following operations are executed, the following Confirm Message is popped up before the memory card program file is modified.



1. Delete folder or program file
2. Rename folder or program file
3. Drop-in program file
4. Add folder

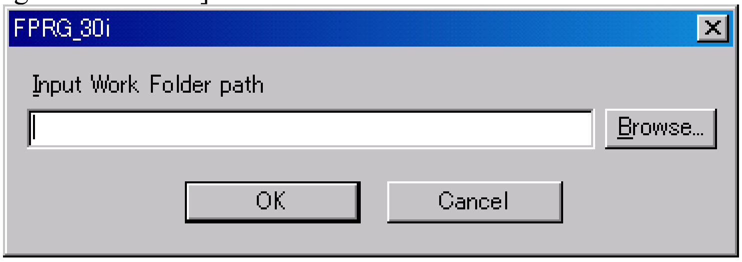
If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

When the [Hide Confirm Message] in menu bar is checked, the Confirm Message is not displayed and the operation advances at once. The default setting is the Confirm Message displayed.

[Ignore Error Code]

When the [Ignore Error Code] in menu bar is checked, program file which includes unusable characters can be dropped in. The unusable characters in the program file will be ignored and will not be written into the memory card program file.

[Change Work Folder]

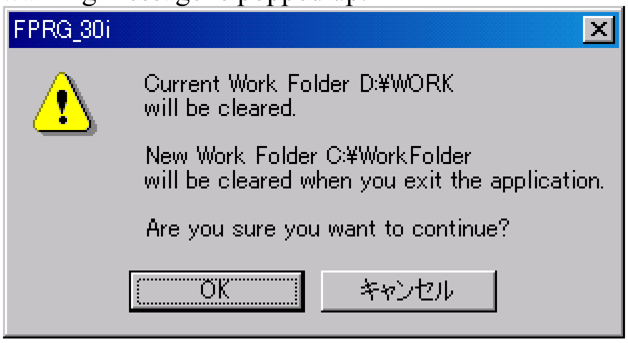


Work folder is used for temporarily keeping the dropped out files. If work folder has no enough free space, Drop-out will not be executed. To avoid this, you can check this option and change the work folder to an area where there is enough free space. With default setting, work folder [temp] will be created on the same place of this PC tool "FPRG_30i.exe".

If you changed the default work folder, the following warning message is popped up.

If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

If the work folder has been changed before, the following warning message is popped up.

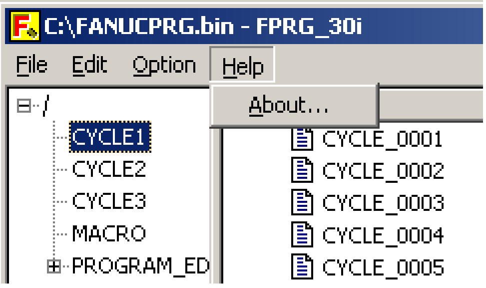


If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

⚠ **WARNING**

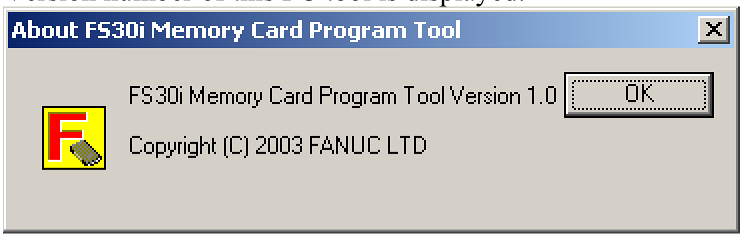
1. At this PC tool terminated, all files in the work folder are deleted.
2. During this PC tool executing, do not access the files in the work folder. If the access is done, there is no assurance of normal operation.

Help menu



[About...]

Version number of this PC tool is displayed.



- Mouse Operation

[Drop-in and Drop-out]

- Drop-in from Explorer

NC program can be added by dropping files including the NC files into the List view window of this PC tool from Explorer

NC program name and update time is the same as the Dropped-in files.

If "Oxxxxxxxx" or "<xxxx>" exists at the top of files, These "Oxxxxxxxx" and "<xxxx>" become NC program name. If not exist, each file name becomes NC program name.

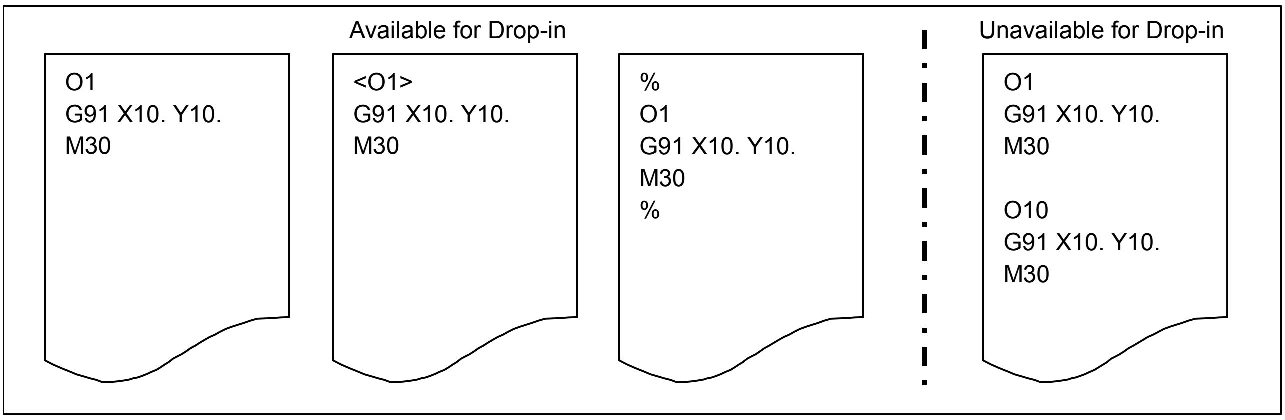
Examples			
Name of external file	**Top of external file**	**Name of internal file**	**Program number**
O1234	N10G00	O1234	1234
O123N10G00	N10G00	O123N10G00	Not an O number program
test.txt	O1234N10G00	O1234	1234
test.txt	<O1234>	O1234	1234
test.txt	<O1234N10>	O1234N10	Not an O number program
O1234	<O1234N10>	O1234N10	Not an O number program
O001234	N10G00	O1234	1234
O001234N10G00	N10G00	O001234N10G00	Not an O number program
test.txt	O001234	O1234	1234
test.txt	<O001234>	O1234	1234
test.txt	<O001234N10G00>	O001234N10G00	Not an O number program

NOTE
1. For naming program file, please refer to the following chapter "Naming rules of Program file".
2. For usable characters in Program file, please refer to the following chapter "Rules of characters in Program file".
3. The updating time of program file is available from 1997 to 2037.

This PC tool can check contents of dropped-in program file according to "Rules of characters in Program file". However, this PC tool does not check grammar of NC program.

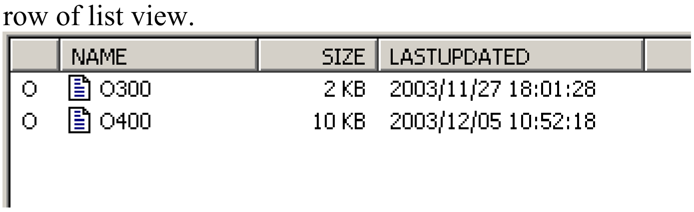
The program file can only have one NC program. Therefore, the NC program output from CNC cannot be dropped in directly.

Example of Program



NOTE
1. If the same named program file exists, another same named program file cannot be dropped in.
2. If free space on the memory card program file is lacking, new program file cannot be dropped in.
3. If the NC program name is not compliant with "Naming rules of Program file", the program file can not be dropped in.

If the File name acts as Program number, "O" is displayed on the first row of list view.



- Drop-out from List view window

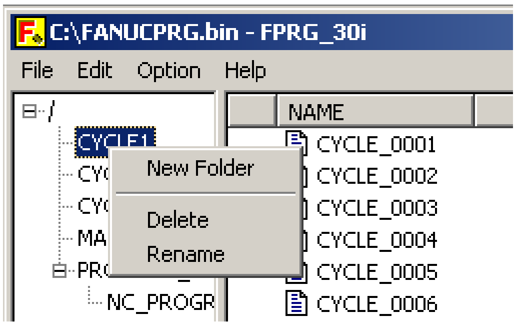
Drop-out from list view of this PC tool to Explorer is available.

> **NOTE**
> Do not drop out to Work folder. If dropped out to Work folder, this PC tool cannot continue to function normally.

- Pop-up menu

Pop-up menu is displayed by clicking the right mouse button.

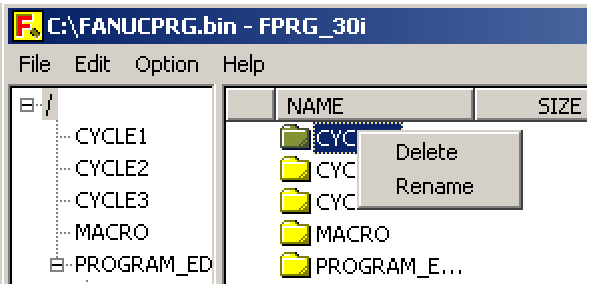
- Focus on Tree view



Clicking "New Folder", a new folder is created on selected folder.
Clicking "Delete", the selected folder is deleted.
Clicking "Rename", the selected folder is renamed.

If clicking on root folder, "Delete" and "Rename" are not activated.

- Focus on List view



Clicking "Delete", the selected folder or program file is deleted.
Clicking "Rename", the selected folder or program file is renamed.

- Display of free space on the memory card program file ("FANUCPRG.BIN")

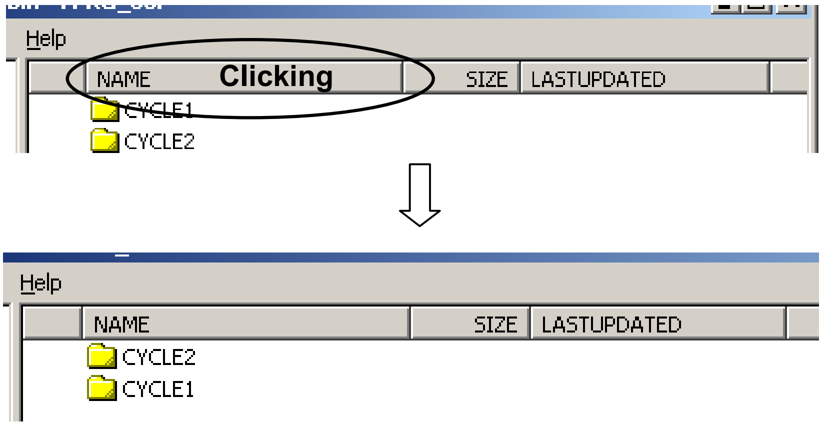
Number of used folder, size of used space, number of unused folder, and size of free space are displayed on status bar in the lower portion of the screen.

When a new memory card program file is created, two reserved folder are also created. Therefore, the number of used folder is two. However it does not show that number of unused folder is reduced.

The display of status bar is renewed at creating or deleting a folder, dropping-in from Explorer, and deleting a program file.

- Sorting list view of the memory card program file

When a column is being clicked, the list view of the memory card program file is being sorted by the column key in ascending or descending order.
The initial display is sorted by NAME key in ascending order.



H.2 NAMING RULES

Overview

Naming rules of folder and program file are described as follows.

H.2.1 Naming Rules of Program File

Here are Naming rules of Program file:
- Program file name can have a maximum of 32 characters.
- Program file name can have following characters.
  Alphabet(Upper and lower case letter), numeric character, "-"(minus), "+"(plus), "_"(under bar), "."(dot)
  "." and ".." can not be used since these are reserved for system use.

- The File name acts as Program number

When the File name is "O"+ 1-99999999, the file name acts as Program number.

Example)

"O123"	Program number 123
"O1"	Program number 1
"O3000"	Program number 3000
"O99999999"	Program number 99999999
"O0123"	Program number 123

The file name does not act as Program number, and can be dropped in:

"ABC"	Top of character is not upper case letter of "O".
"o123"	Top of character is not upper case letter of "O".
"O0123XY"	Characters following the "O" contains alphabet.

The file name does not act as Program number, and cannot be dropped in:

"O123456789"	Numeric characters exceed 8 digits.

> **NOTE**
> 1 Program file name cannot be repeated in a Folder.
> 2 If program file name starts with "O" and the next eight characters are all numeric characters, the "0" (zero) after the "O" will be deleted.

H.2.2 Naming Rules Of Folder

Here are Naming rules of Folder:
- Folder name can have a maximum of 32 characters.
- Folder name can have following characters.
  Alphabet(Upper and lower case letter), numeric character,
  "-"(minus), "+"(plus), "_"(under bar), "." (dot)
  "." and ".." can not be used since these are reserved for system use.

NOTE
Folder name cannot be repeated in a Folder.

H.3 RULES OF CHARACTERS IN PROGRAM FILE

Overview

Words in parentheses "( )" in Program file are treated as comments.
The mark of comment start "(" is named "Control-out".
The mark of comment end ")" is named "Control-in".
"Control-out" and "Control-in" must make a pair. The order is 1st - "Control-out" and 2nd -"Control-in". And nested parentheses is not available.

NOTE

1 When a program file is dropped in, space code (0x20 SPC) , tab code (0x09 HT) , carriage return code(0x0d CR) and percent code(0x25 %) is deleted. If "%" is found in Control-in, characters between "%" and next "LF" (0x0a) is deleted.
2 The front of program number ":" is changed to "O"(O as in Oscar) while the program file dropped in.
3 The program file can only have one NC program.

H.3.1 Usable Characters in Program File

- Usable characters in Control-in

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
0a	LF	3f	?	58	X	74	t
23	#	40	@	59	Y	75	u
26	&	41	A	5a	Z	76	v
28	(	42	B	5b	[	77	w
29	)	43	C	5d	]	78	x
2a	*	44	D	5f	_	79	y
2b	+	45	E	61	a	7a	z
2c	,	46	F	62	b		
2d	-	47	G	63	c		
2e	.	48	H	64	d		
2f	/	49	I	65	e		
30	0	4a	J	66	f		
31	1	4b	K	67	g		
32	2	4c	L	68	h		
33	3	4d	M	69	i		
34	4	4e	N	6a	j		
35	5	4f	O	6b	k		
36	6	50	P	6c	l		
37	7	51	Q	6d	m		
38	8	52	R	6e	n		
39	9	53	S	6f	o		
3a	:	54	T	70	p		
3c	<	55	U	71	q		
3d	=	56	V	72	r		
3e	>	57	W	73	s		

NOTE
In the Control-in, "O", ":", and "<" can not be used at top of the line except for the 1st line.

- Usable characters in Control-out(characters in parentheses)

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
0a	LF	3c	<	55	U	71	q
20	SPC	3d	=	56	V	72	r
22	“	3e	>	57	W	73	s
23	#	3f	?	58	X	74	t
24	$	40	@	59	Y	75	u
26	&	41	A	5a	Z	76	v
27	‘	42	B	5b	[	77	w
2a	*	43	C	5d	]	78	x
2b	+	44	D	5f	_	79	y
2c	,	45	E	61	a	7a	z
2d	-	46	F	62	b		
2e	.	47	G	63	c		
2f	/	48	H	64	d		
30	0	49	I	65	e		
31	1	4a	J	66	f		
32	2	4b	K	67	g		
33	3	4c	L	68	h		
34	4	4d	M	69	i		
35	5	4e	N	6a	j		
36	6	4f	O	6b	k		
37	7	50	P	6c	l		
38	8	51	Q	6d	m		
39	9	52	R	6e	n		
3a	:	53	S	6f	o		
3b	;	54	T	70	p		

H.4 ERROR MESSAGE AND NOTE

Error may occur when using this application, hereafter explains the error messages and gives relative instructions.

H.4.1 List of Error Message

When an error occurred, the error message box is displayed as follows.

Message	Remarks
Failed to open the file you specified.	If failed to open once again, the file may be broken.
Failed to read or write to the specified file.	
There is insufficient disk space.	There is no enough free space to create a new memory card program file or to put drop out files to work folder. In the latter case, please refer to chapter "Menu" [Change Work Folder]
File name is not correct.	Please refer to the chapter "Naming rules of Program file".
Input name is already exists.	Please input another name.
File name is already exists.	Please input another name.
Input name is not correct.	Please refer to the chapter "Naming rules".
Please input an integer between 2 and 2048	The size of the memory card program file is available from 2Mbyte to 2048Mbyte.
An illegal character is included in the specified file.	Please refer to the chapter "Rules of characters in Program file".
Last update time of the specified file is unsupported.	The updating time of program file is available from 1997 to 2037.
The memory card program file you specified cannot be identified.	The specified file format is not the memory card program file
There is insufficient free folder.	Please delete unnecessary program files or folders.
There is insufficient free program space.	Please delete unnecessary program files.
Folder deeper than 7 cannot be created.	
Root folder cannot be deleted.	
Root folder cannot be renamed.	
Failed to create work folder.	Please check whether the [temp] work folder is able to create or not. In default, the [temp] work folder is created on the same place of this PC tool "FPRG_30i.exe".
Process has been cancelled.	
The specified work folder is not found.	The Drop-out cannot be executed. Terminate this PC tool. And check the setting of Work Folder on the Option dialogue window.
Only one instance of this application can be executed.	Double activation of this PC tool cannot be executed.

H.4.2 Note

- Folder and Program Numbers

Folder and Program Numbers can be selected among 63 / 500 / 1000.
For selecting the Numbers of 500 or 1000, the option "Registered programs expan. On the memory card" is needed in CNC side.

INDEX

<N>

<O>

<P>

<R>

<S>

Revision Record

FANUC Series 30*i*/300*i*/300*i*s-MODEL A, Series 31*i*/310*i*/310*i*s-MODEL A5, Series 31*i*/310*i*/310*i*s-MODEL A, Series 32*i*/320*i*/320*i*s-MODEL A USER'S MANUAL (Common to Lathe System/Machining Center System)(B-63944EN)

Edition	Date	Contents	Edition	Date	Contents
02	Jun, 2004	Addition of functions Addition of following models - Series 31*i* /310*i* /310*i*s-MODEL A5 - Series 31*i* /310*i* /310*i*s-MODEL A - Series 32*i* /320*i* /320*i*s-MODEL A			
01	Jul., 2003	————————			